



UNITED STATES PATENT AND TRADEMARK OFFICE

2123
JW
I
UNITED STATES DEPARTMENT OF COMMERCE
United States Patent and Trademark Office
Address: COMMISSIONER FOR PATENTS
P.O. Box 1450
Alexandria, Virginia 22313-1450
www.uspto.gov

APPLICATION NO.	FILING DATE	FIRST NAMED INVENTOR	ATTORNEY DOCKET NO.	CONFIRMATION NO.
10/064,629	07/31/2002	Calvin Edward Phillips	000031562-1	8858
31562	7590	02/27/2006	EXAMINER	
APPLIEDVB INC. 359 SPODE WAY SAN JOSE, CA 95123			SHARON, AYAL I	
			ART UNIT	PAPER NUMBER
			2123	
DATE MAILED: 02/27/2006				

Please find below and/or attached an Office communication concerning this application or proceeding.

SAP2000®

Linear and Nonlinear
Static and Dynamic
Analysis and Design
of
Three-Dimensional Structures

INTRODUCTORY TUTORIAL



Computers and Structures, Inc.
Berkeley, California, USA

Version 8.0
June 2002

Copyright

The computer program SAP2000 and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

Phone: (510) 845-2177
FAX: (510) 845-4096
e-mail: info@csiberkeley.com (for general questions)
e-mail: support@csiberkeley.com (for technical support questions)
web: www.csiberkeley.com

© Copyright Computers and Structures, Inc., 1978-2002.
The CSI Logo is a registered trademark of Computers and Structures, Inc.
SAP2000 is a registered trademark of Computers and Structures, Inc.
Windows is a registered trademark of Microsoft Corporation.
Adobe and Acrobat are registered trademarks of Adobe Systems Incorporated.

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Contents

Introductory Tutorial for SAP2000

1	Introduction	
	Using This Manual	1-1
	Overview of the Program	1-2
	Using this Tutorial	1-2
2	An Introductory Tutorial	
	The Project	2-2
	The Interface	2-2
	Step 1 Begin a New Model	2-3
	Define an Auto Select Section	
	List	2-6
	Step 2 Add Frame Objects	2-10
	Draw Frame Objects	2-10
	Replicating Objects	2-11
	Trimming Objects	2-14
	Assigning Member End Releases	2-17
	Save the Model	2-19
	Step 3 Add Area Objects	2-19
	Define the Area Sections	2-19
	Draw the Area Object	2-20
	Mesh the Area Object	2-22
	Step 4 Add Restraints	2-23

Introductory Tutorial for SAP2000 Version 8

Step 5	Define Load Cases	2-25
Step 6	Assign Gravity Loads	2-26
Step 7	Assign Area Stiffness Modifiers	2-28
Step 8	Run the Analysis	2-29
Step 9	Graphically Review the Analysis Results	2-30
Step 10	Design the Steel Frame Objects	2-34

Chapter 1

Introduction

Using this Manual

This manual introduces you to SAP2000 Version 8. The step-by-step instructions guide you through development of your first model. The intent is to demonstrate the fundamentals and to show how quickly and easily a model can be created using this program. This tutorial is intended to give you hands-on experience working with SAP2000, which for most people, is the quickest way to become familiar with the program.

SAP2000 is an extremely versatile and powerful program with many features and functions. This manual does not attempt to fully document all of those capabilities. Rather, we briefly show how to work with the program, providing some commentary along the way. To grasp the full value of SAP2000, you should use this introductory tutorial manual in conjunction with the other SAP2000 documentation.

We hope that you enjoy using this tutorial, and that you find it beneficial as a starting point in your exploration of this powerful and comprehensive version of SAP2000.

1

Overview of the Program

SAP2000 is a stand-alone finite-element-based structural program for the analysis and design of civil structures. It offers an intuitive, yet powerful user interface with many tools to aid in the quick and accurate construction of models, along with the sophisticated analytical techniques needed to do the most complex projects.

SAP2000 is object based, meaning that the models are created with members that represent the physical reality. A beam with multiple members framing into it is created as a single object, just as it exists in the real world, and the subdividing needed to ensure that connectivity exists with the other members is handled internally by the program. Results for analysis and design are reported for the overall object, and not for each sub-element that makes up the object, providing information that is both easier to interpret and more consistent with the physical structure.

Using this Tutorial

The example in this tutorial provides a step-by-step description of how to use SAP2000. We recommend that you actually perform these steps in SAP2000 while reading this manual.

The SAP2000 program must be installed on your computer before you can begin the tutorial. It would also be a good idea to peruse the other SAP2000 documentation prior to starting this tutorial, or at least have them readily available if needed.

If you are viewing this tutorial manual as a .pdf file, we strongly recommend that you print it out before starting the tutorial. It will not be practical to use the SAP2000 program while trying to read this manual on your computer screen.

During the course of this tutorial, we will explore many of the basic features of SAP2000. Prepare to spend at least one hour going through this example, and if at any time you need to stop, save your model so that you may continue at a later time.

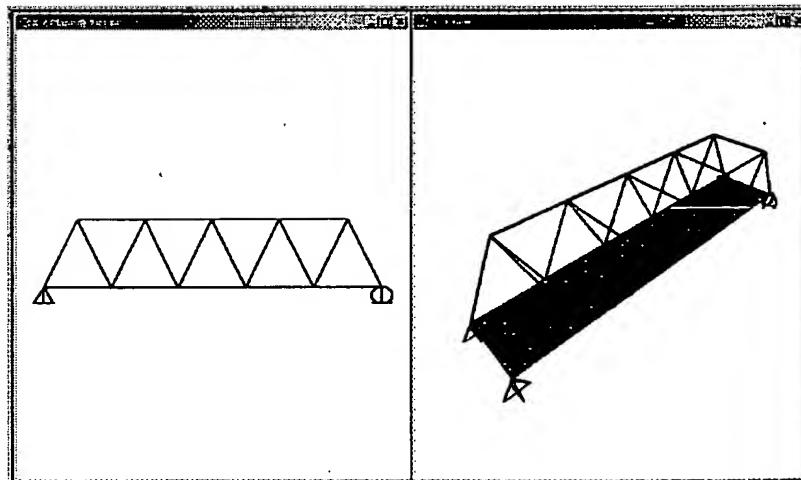
Welcome to SAP2000.

Chapter 2

An Introductory Tutorial

This chapter provides step-by-step instructions for building a basic SAP2000 model. Each step of the model creation process is identified, and various model construction techniques are introduced. At the completion of this chapter, you will have built the model shown in Figure 1.

Figure 1
*The Tutorial
Model*



The Project

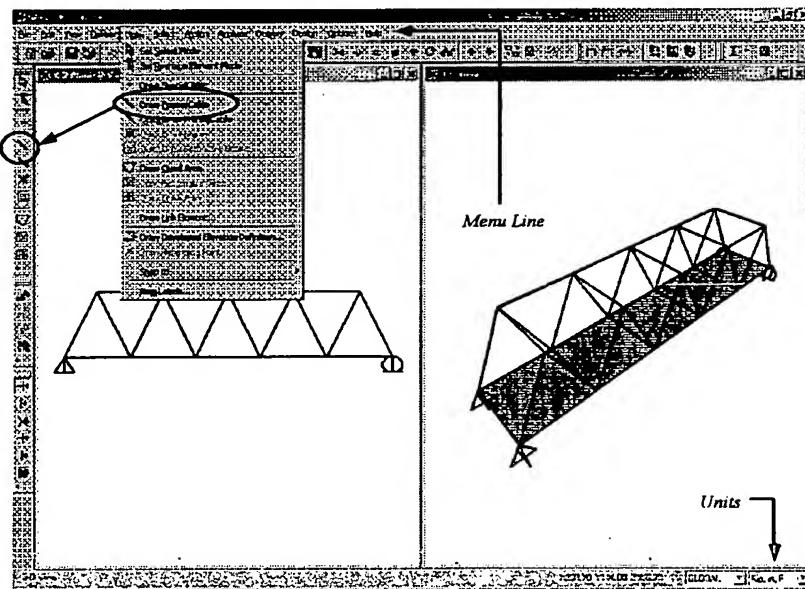
The tutorial project is a five panel, sloped truss bridge. The bridge spans 60 feet, and has a width and height of 12 feet each. The supports are rollers at one end, and pins at the other.

The trusses and cross members are to be constructed of 2L4X4's, while the deck will be a concrete slab 5 inches thick. The bridge will be analyzed for static loads only, and the deck will be loaded with a Dead Load = 10 pounds per square foot (psf) and a Live Load = 100 psf.

The Interface

The top menu line contains all of the commands and options available to SAP2000, including Define, Draw, Select, Assign, Analyze, Display and Design. These listed menus contain the commands that will be needed most often when using SAP2000, and many of the most frequently used commands are accessible as a single click button in the screen regions surrounding the drawing areas. The availability of a button is indicated in the main menus by the existence of an icon to the left of the command. The lower right corner shows the current unit selection. Figure 2 shows the layout of the interface.

Figure 2
The Interface

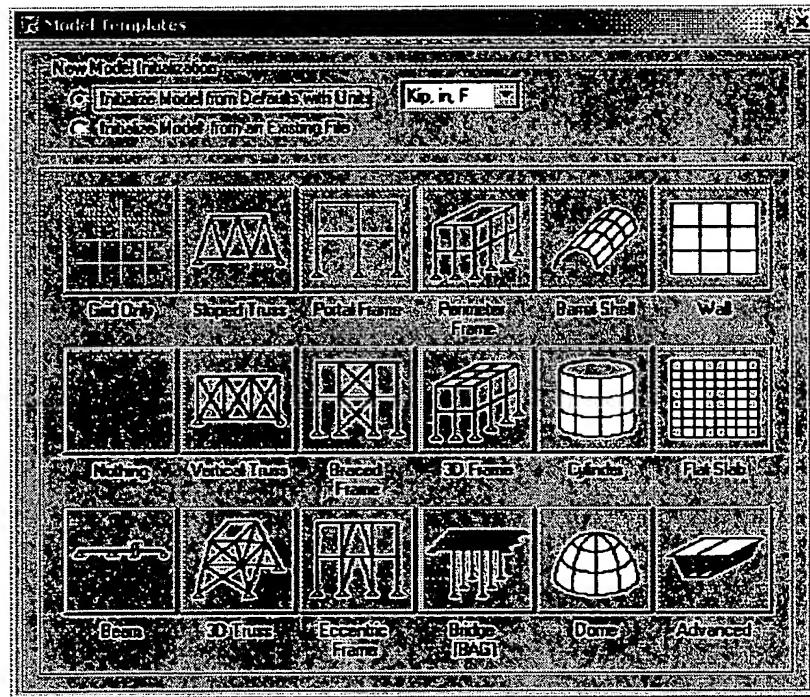


Step 1 Begin a New Model

In this Step, the basic grid will be defined which will serve as a template for developing the model. Then a list of double angle sections will be selected for the truss Auto Select list.

- Click the File menu > New Model command or the New Model button . The form shown in Figure 3 will display. Verify that the default units are set to Kip, in, F.

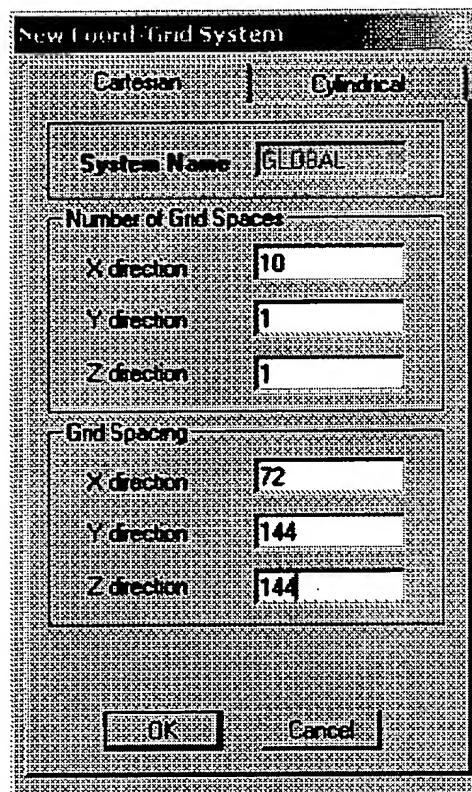
Figure 3
Model
Templates



- The Model Template form allows for the quick generation of numerous model types using parametric generation techniques. However, in this tutorial the model will be started using only the grid generation. When laying out the grid, it is important that the geometry defined accurately represents the major geometrical aspects of the

model, so it is advisable to spend time carefully planning the number and spacing of the grid lines. Select the **Grid Only** button, and the form shown in Figure 4 will display.

Figure 4
New Coord/Grid
System form



C. The New Coord/Grid System form is used to specify the grids and spacing in the X, Y and Z direction. Set the number of grid spaces to 10 for the X direction, and to 1 for the Y and Z directions. Type 6 ft into the X direction spacing edit box and press the Enter key on your keyboard. Note that the program automatically converts the 6 ft to 72 to be consistent with the default units of inches. Enter 12 ft or 144 for both the Y and Z direction spacing.

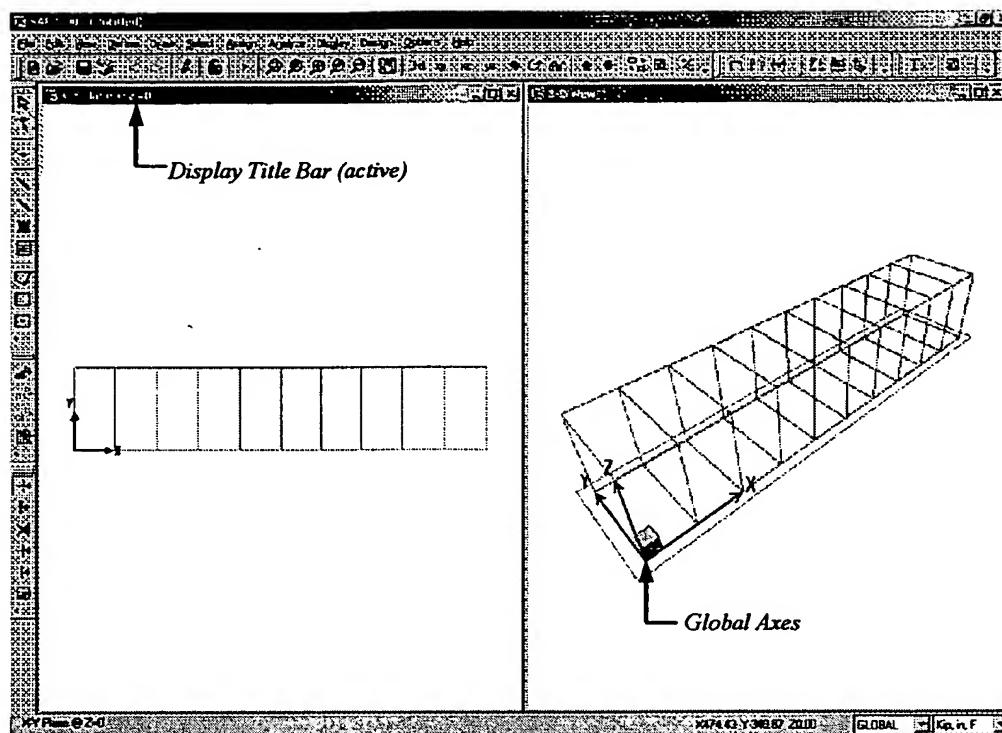


Figure 5
The SAP2000
windows

D. Click the **OK** button to accept the changes, and Figure 5 will appear.

The grids appear in two view windows tiled vertically, a X-Y “Plan” View on the left and a 3-D View on the right, as shown above. The number of view windows may be changed by selecting the **Options** menu > **Windows** command.

Notice that the “Plan” view is active in Figure 5. When the window is active, the display title bar is highlighted. Set a view active by clicking anywhere in the view window.

Note that the Global Axes are displayed as well, and that Z positive is in the “up” direction. When SAP2000 refers to the direction of gravity, this is in the negative Z direction, or “down”.

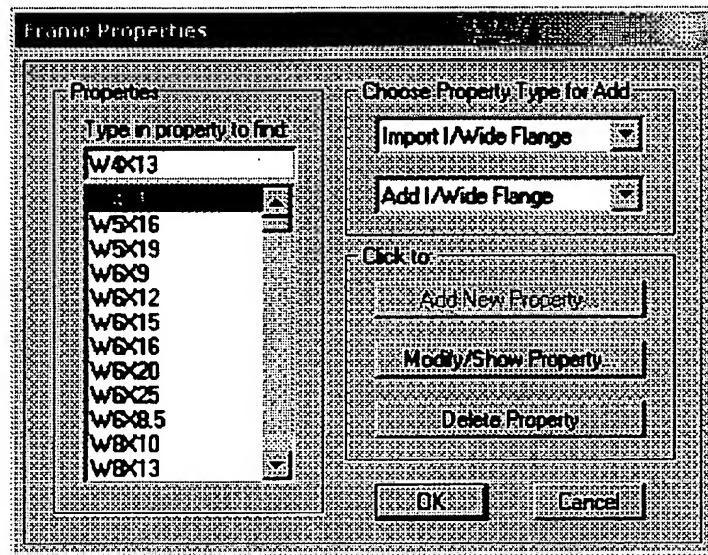
Define an Auto Select Section List

An auto select section list is simply a list of sections, which for this tutorial will be a set of double angles (2L4X4's). Auto select section lists are assigned to frame objects in the same manner as an individual section property. When an auto select section list is assigned to a frame object, the program can automatically select the most economical, adequate section from the list when designing the member. When performing the initial analysis, the program will assign the median section from the list for the analysis properties.

For this particular tutorial, the program will analyze and design from a set of double angles, which will be chosen from an auto select sections list created now.

- A. Click the **Define** menu > **Frame/Cable Sections** command, which will display the Frame Properties form shown in Figure 6.

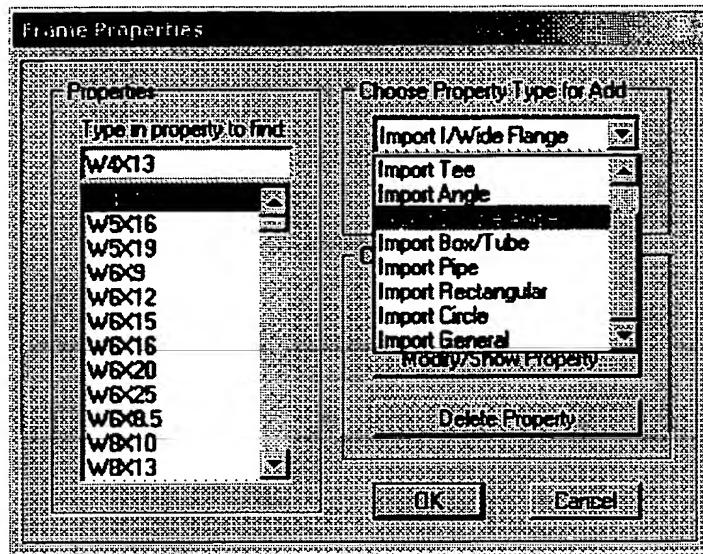
Figure 6
The Frame
Properties
form



- B. Scroll down the sections listed under Properties to see if the list contains 2L4X4's, and if it does, skip ahead to Step D. Otherwise, proceed to Step C.

C. Click the drop-down box that reads “*Import I/Wide Flange*” in the Choose Property Type for Add area of the form. Scroll down the list of import options until you find *Import Double Angle* – see Figure 7. Single click on it.

Figure 7
Import
Double
Angle



D. In the Click to area of the Frame Properties form, click the Add New Property button, which will open the Section Property File form.

E. Select and open the file named *SECTIONS8.PRO* from the Section Property File form, as this file contains the properties of the double angles to be used in the model. The Sections8.pro sections list form shown in Figure 8 appears.

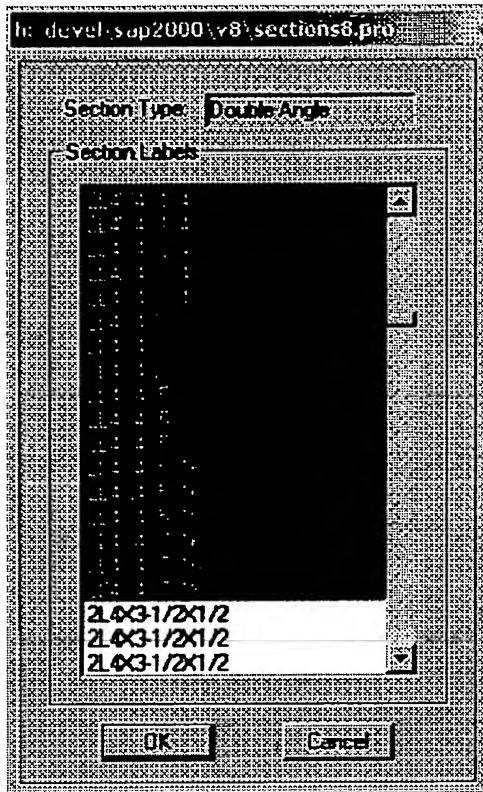
F. Scroll down the list of double angles in the Sections Labels area until you find the first 2L4X4. Click once on that member to highlight it.

G. Scroll further down the list until you find the last 2L4X4. Hold down the Shift key on your keyboard and click once on the last 2L4X4X7/16 – all of the 2L4X4’s should now be highlighted.

2

H. Click the **OK** button, and then click the **OK** button in the Double Angle Section form to add the angles selected to the list in the Properties area on the Frame Properties form.

*Figure 8
Sections8.pro
sections
list*

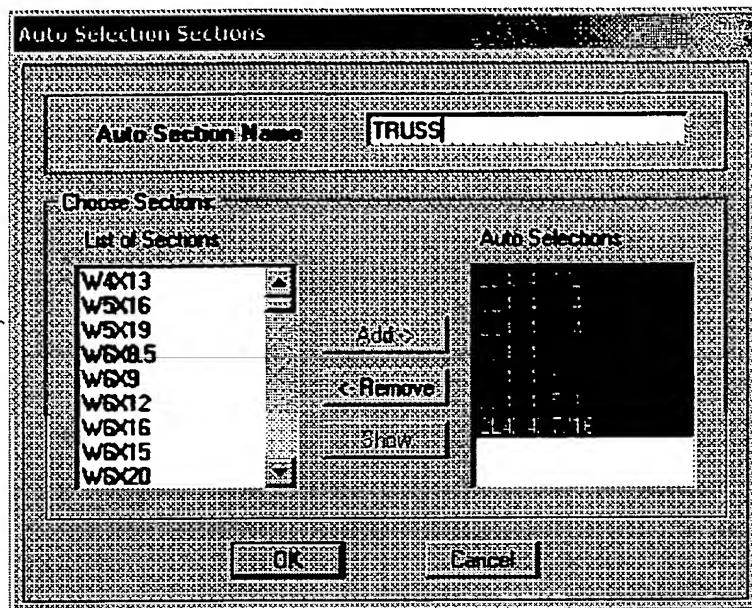


I. Click the drop-down box that reads "Add I/Wide Flange" in the Choose Property Type for Add area of the Frame Properties form and scroll down until you locate *Add Auto Select*. Single click on it.

J. In the Click To area of the Frame Properties form, click the **Add New Property** button, which will open the Auto Selection Sections form shown in Figure 9.

K. Type **TRUSS** in the Auto Section Name edit box.

Figure 9
Auto
Selection
Sections
form



- L. Scroll down the List of Sections to find the $2L4X4X1/2$ double angle, and click once to highlight it.
- M. Continue down the list until you find the last double angle, $2L4X4X7/16$, and while holding down the shift key on the keyboard, click once on this section. All of the $2L4X4$'s should now be highlighted.
- N. Click the Add button to move the selected list to the Auto Selections edit box on the right side of the form.
- O. Click the OK button and then click the OK button on the Frame Properties form to accept your changes and add the TRUSS auto select list to the Properties edit box.

2

Step 2 Add Frame Objects

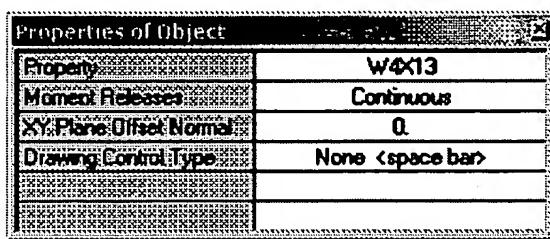
In this Step, Frame objects with the associated TRUSS sections list are drawn using the grids and snap-to options, and generated with Edit menu commands.

Draw Frame Objects

Make sure that the X-Y Plane @ Z=0 view (plan at lowest elevation) is active (see Step1-D for directions on how to make a view active and page 2-14 for setting the view). This view should be in the left window. Also check that the **Snap to Points** and **Grid Intersections** command is active. This will assist in accurately positioning the frame objects. This command is active when its associated button  is depressed. Alternatively, use the **Draw menu > Snap to > Points and Grid Intersections** command. By default, this command is active.

- A. Click the **Draw Frame/Cable**  button or use the **Draw menu > Draw Frame/Cable** command. If you accessed the Draw Frame/Cable command via the Draw menu, the Draw Frame/Cable button will depress verifying your command selection. The Properties of Object pop-up box for frames will appear as shown in Figure 10.

*Figure 10
Properties of
Object box*



If the Properties of Object box is covering any part of the model in either view, drag it out of the way.

- B. Click in the Property edit box on the Properties of Object form and scroll down to **TRUSS**. Single click on it to assign the auto select list **TRUSS** to the members you will draw.

C. To draw the first frame object, left click once in the X-Y Plane view at the X-Y origin, and then click again at the far right end along the same horizontal grid line ($x=720, y=0$). The cursor location is indicated in the lower right-hand corner of the interface. A frame line should appear in both views (plan and 3D). After clicking to define the end point of the frame object, a right click will "lift the pen" so you will no longer be actively drawing, but will leave the Draw Frame/Cable command active so that you may add additional elements.

If you have made a mistake while drawing this object, click the **Select Object**  button, to leave the Draw mode and go to the Select mode. Then click the **Edit menu > Undo Frame Add** command, and repeat Items A-C.

D. Repeat Item C, drawing an additional frame object parallel to the first member from ($x=0, y=144$) to ($x=720, y=144$). These members form the bottom chords of the trusses. Right click to stop drawing.

E. Left click at ($x=0, y=0$) and then at ($x=0, y=144$) to draw the first transverse member.

F. Click on the **Select Object**  button, or Press the Esc key on the keyboard to exit the Draw Frame/Cable command.

Replicating Objects

Make sure that the program is in the Select mode.

A. Select the transverse member spanning between the longitudinal chords by a left click directly on the member, or left click to the right of the object, and while holding the left mouse button down, drag the mouse across the member. See Figure 12 for selection options.

B. Click the **Edit menu > Replicate** command to bring up the form shown in Figure 11.

C. On the Linear tab, type 144 into the dx edit box.

D. Type 5 into the Number edit box.

2

Figure 11
Replicate
box

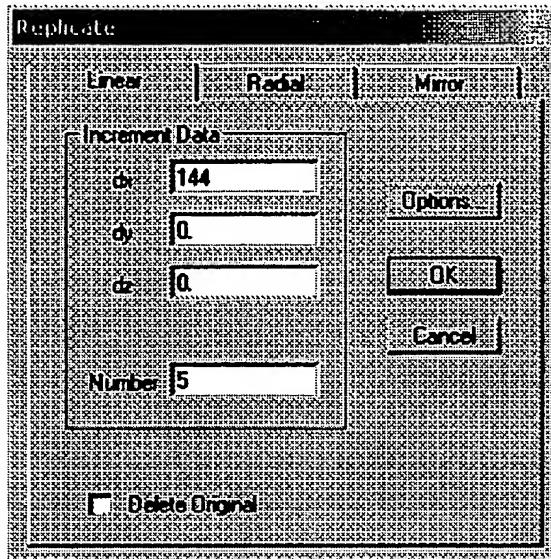
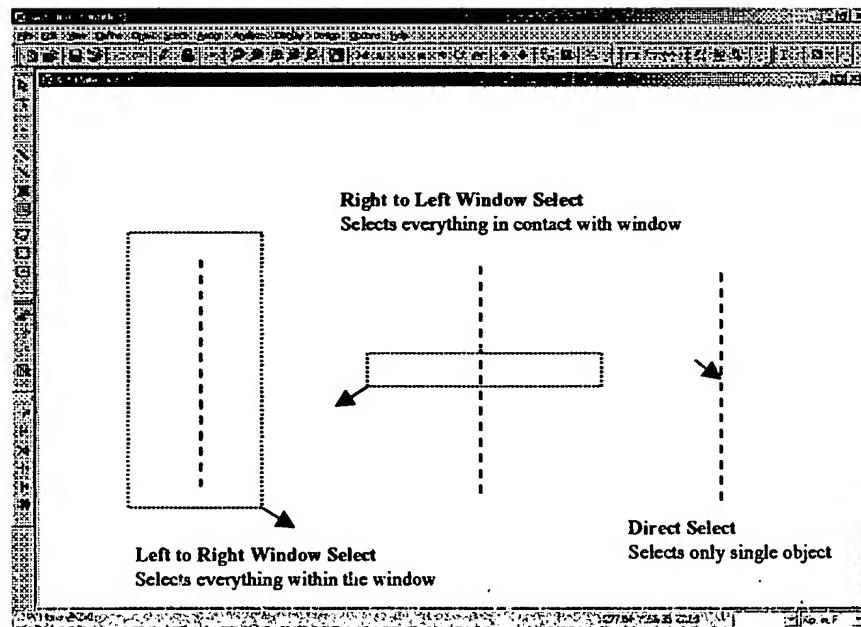


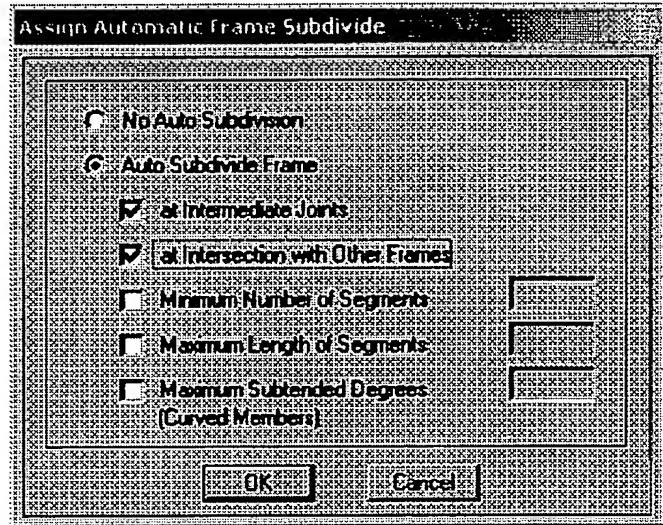
Figure 12
Graphical
Selection
Options



- F. Left click once on each of the longitudinal chord members to select them.
- G. Click on the **Assign** menu > **Frame/Cable** > **Automatic Frame Subdivide** command to bring up the form in Figure 13. Select the *Auto Subdivide Frame* option and check the *at Intermediate Joints* and *at Intersection with Other Frames* check boxes, and click **OK**.

This subdivision is necessary to ensure connectivity between the chords and the other members because the chords were drawn as single “physical” objects. From an analytical standpoint, the chords will now be connected to all of the elements framing into them, but for design and selection they will remain as single objects.

Figure 13
Assign Automatic Frame Subdivide form



- H. Click the **Select All**  button or use the **Select** menu > **Select > All** command to select all of the objects currently in the model.
- I. Click the **Edit** menu > **Replicate** command to bring up the Replicate form.
 - 1. Type **72** into the **dx** edit box, **0** into the **dy** box, and **144** into the **dz** box.

2. Type 1 into the Number edit box.

3. Click **OK** to accept the changes.

2

The framing at the bottom plan will be replicated at the top level with a shift of 72 inches in the X direction.

Trimming Objects

Make sure that the program is in the select mode, and that the X-Y view is active.

A. Click the **View** menu > **Set 2D View** command

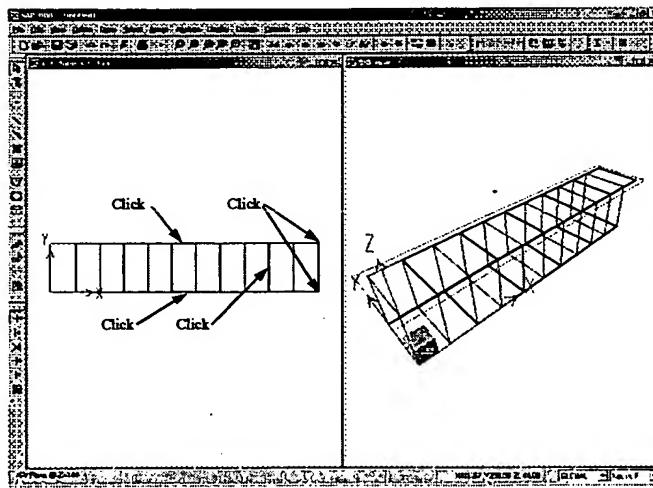
1. In the Set 2D View form click on the *X-Y plane* option.

2. Type 144 into the *Z=* edit box to display the plan view at the upper elevation.

B. Click the **Assign** menu > **Clear Display of Assigns** command to remove the Frame Subdivide identifiers.

C. Click on both top chords, the next to last transverse member to the right, and the two point objects at the far right ends of both chords, as shown in Figure 14. The selected objects should be shown as dashed lines.

Figure 14
Select
mode for
Trim



D. Click the **Edit menu > Trim/Extend Frames** command, which brings up the Trim/Extend Selected Frames form.

1. Select the *Trim Frames* option, and click **OK**.

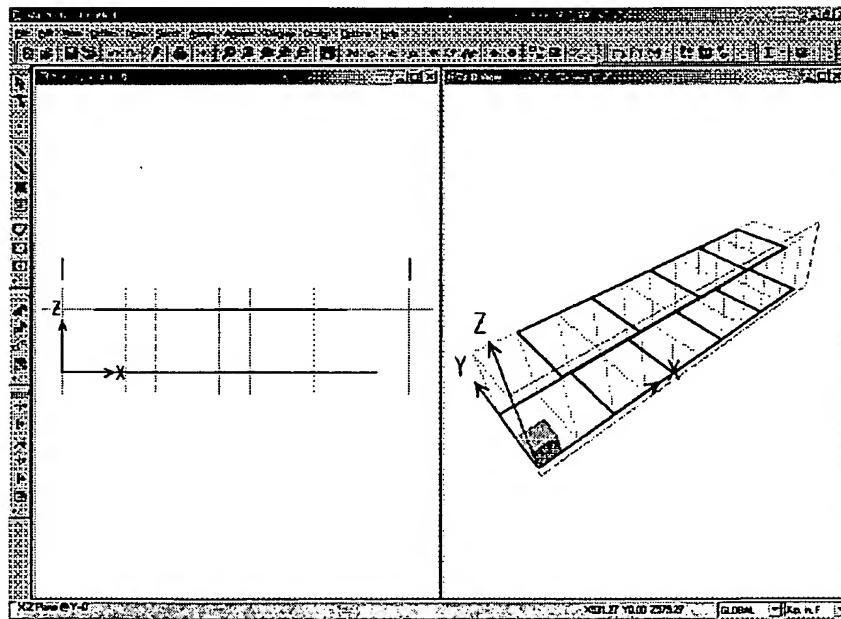
Selecting the Trim Frames option will trim the two top chords beyond the next to last transverse member. To trim a Frame member, select the member, select a member to be used as the trim location, and select a point object on the side to be trimmed.

E. Click on the “orphaned” transverse frame member on the far right, and go to the **Edit menu > Delete** command, or Press the **Delete** key on your keyboard.

F. Make sure that the plan view is active and click the **XZ View** button.

Your model now appears as shown in Figure 15.

Figure 15
Model after
frame ob-
jects have
been added
in plan



2

- G. Click the **Draw Frame/Cable**  button or use the **Draw menu > Draw Frame/Cable** command. The **Properties of Object** pop-up box for frames will appear.
- H. Make sure that the **Property** item on the **Properties of Object** form is set to **TRUSS**.
- I. To draw the first diagonal, left click once in the X-Z Plane view at the X-Z origin, and then click again at the nearest end of the top chord ($x=72, z=144$). Without clicking on the right mouse button, add a second diagonal by doing a left click at point ($x=144, z=0$).

Diagonals for one bay are now drawn.

- J. Right click and then click on the **Select Object**  button, or Press the Esc key on the keyboard to exit the **Draw Frame/Cable** command.
- K. Draw a Selection Box from Right to Left across the two diagonals just drawn to select both diagonals. See figure 12 for selection options.
- L. Click the **Edit > Replicate** command to bring up the **Replicate** form.
 1. Type **144** into the **dx** edit box, **0** into the **dy** box, and **0** into the **dz** box.
 2. Type **4** into the **Number** edit box.
 3. Click **OK** to accept the changes.

All of the diagonals for one truss have been drawn.

- M. Draw a Selection Box from Right to Left across all of the diagonals.
- N. Click the **Edit > Replicate** command to bring up the **Replicate** form.
 1. On the **Linear** tab, type **0** into the **dx** edit box, **144** into the **dy** box, and **0** into the **dz** box.
 2. Type **1** into the **Number** edit box.

3. Click **OK** to accept the changes.

The model now appears as shown in Figure 16.

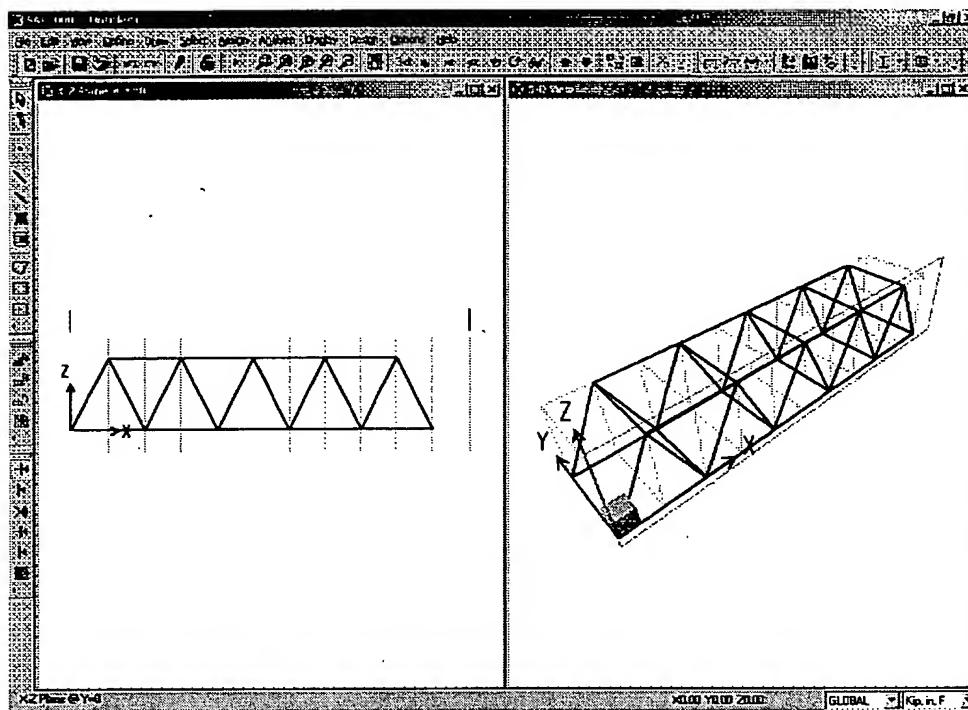


Figure 16
Model after all frame
objects have been added

Assigning Member End Releases

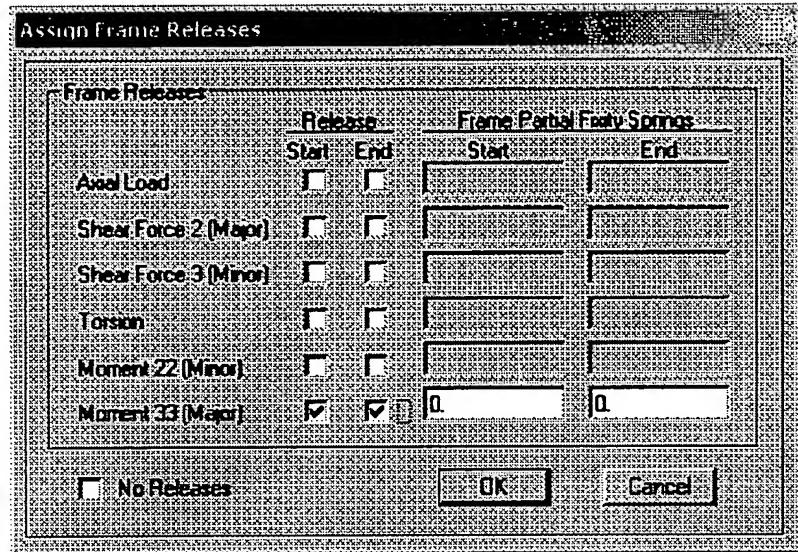
Make sure that the program is in the select mode, and that the X-Z view is active.

- A. Draw a Selection Box from Right to Left across all of the diagonals.
- B. Click on the **Assign** menu > **Frame/Cable** > **Releases/Partial Fixity** command to bring up the form shown in Figure 17. Check the **Moment 33 (Major)** check boxes for both the *Start* and *End* Releases.

By releasing the moments in the major direction, the diagonals in the trusses will behave as pinned elements.

2

Figure 17
Assign Frame
Releases
form



- C. Click **OK** to accept the changes and return to the select mode.
- D. Click the **View menu > Set 2D View** command. In the Set 2D View form click on the *X-Z plane* option and type **144** into the *Y=* edit box to display the second elevation view. Alternatively, use the **Move Up in List** button to toggle to the other elevation.
- E. Draw a Selection Box from Right to Left across all of the diagonals.
- F. Click on the **Assign menu > Frame/Cable > Releases/Partial Fixity** command to bring up the Assign Frame Releases form and make sure that the *Moment 33 (Major)* check boxes for both the *Start* and *End* Releases are checked. Click **OK** to accept changes.
- G. Click the **Assign menu > Clear Display of Assigns** command to remove the Frame Releases identifiers.

Save the Model

During development, save the model often. Although typically you will save it with the same name, on occasion you may want to save it with a different name to record your work at various stages of development.

- A. Click the **File** menu > **Save** command, or the **Save**  button, to save your model. Specify the directory in which you want to save the model and, for this tutorial, specify the file name **Truss**.

Step 3 Add Area Objects

In this step, a concrete deck is added to the model.

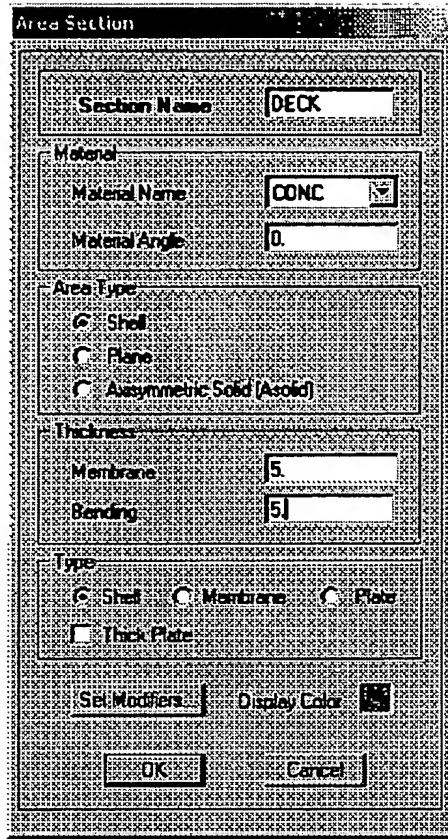
Define the Area Sections

Make sure that the **X-Z** view is active. Now switch to a “plan” view, and define the properties for the concrete deck.

- A. Click the **XY View**  button to display the lowest plan view.
- B. Click the **Define** menu > **Area Sections** command. The **Area Sections** form will appear.
- C. Click the **Add New Sections** button in the **Click to area** of the form. The **Area Section** form shown in Figure 18 appears.
 1. Type **DECK** into the **Section Name** edit box.
 2. Set the **Thickness** (both **Membrane** and **Bending**) to **5** to indicate that the concrete deck is **5** inches thick.

By definition, a **Shell** object has both **Membrane** and **Bending** behavior.
 3. Click the **OK** button and then click the **OK** button in the **Area Sections** form to accept your changes.

Figure 18
Area Section
form



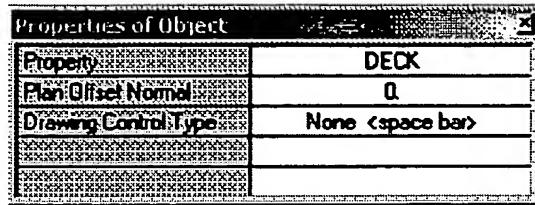
Draw the Area Object

Make sure that the X-Y Plane @ Z=0 view is active. Now draw an area object to represent the deck using the following Action Items.

- Click the **Draw Quad Area Element** button, or go to the **Draw** menu > **Draw Quad Area** command. The Properties of Object pop-up box for areas will appear as shown in Figure 19.

Make sure that the **Property** item in this box is set to *DECK*. If it is not, click once in the edit box opposite the **Property** item to activate the drop-down menu and select *DECK* from the list.

Figure 19
Properties of
Object box



- B. Check that the **Snap to Points and Grid Intersections** command is active. This will assist in accurately drawing the area object.
- C. Click once at point (x=0,y=0). Then moving clockwise around the model, click once at these object points in this order to draw the outline of the deck: (x=0,y=144), (x=720,y=144) and (x=720,y=0).
- D. Click on the **Select Object** button, or Press the Esc key on the keyboard to exit the Draw Quad Area command.
- E. To better view the deck addition, click the **Set Display Options** button. When the form appears, check the *Fill Objects* check box and the *Apply to All Windows* check box, as shown in Figure 20.

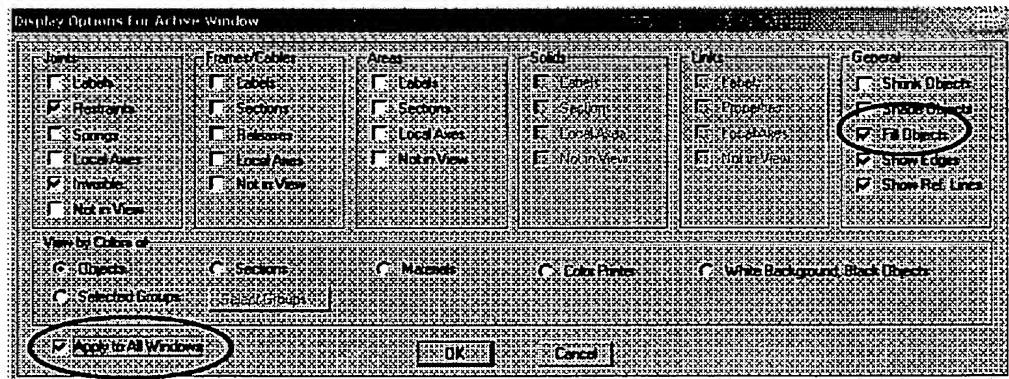


Figure 20
Display Options for
Active Windows form

- F. Click **OK** to accept the changes, and the model now appears as shown in Figure 21.

2

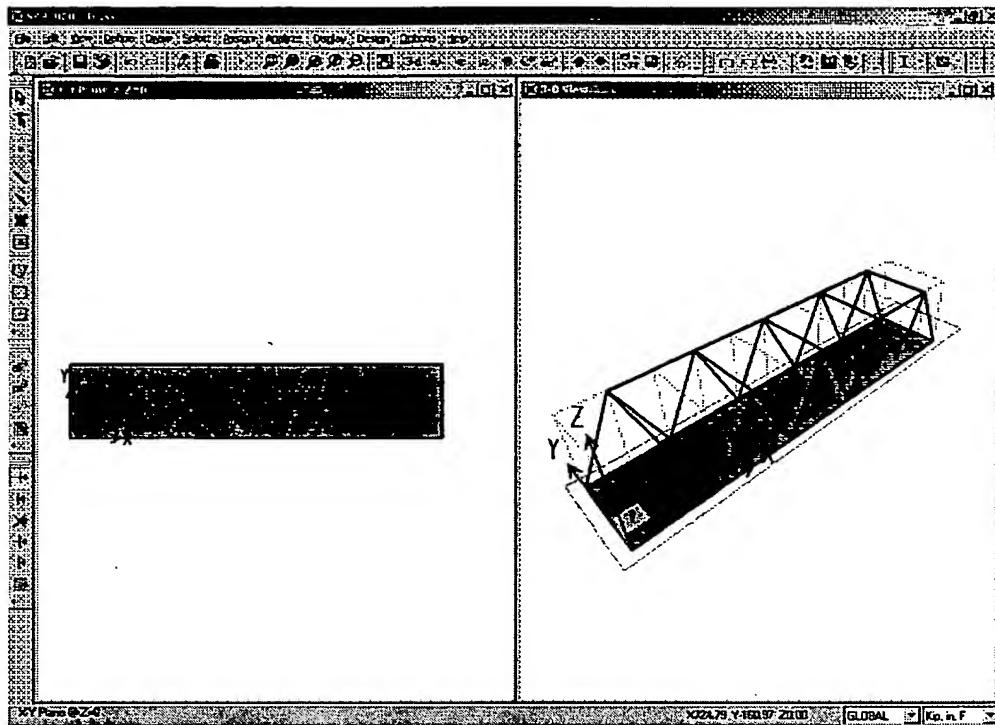


Figure 21
Model after the area
objects have been drawn

Mesh the Area Object

Make sure that the X-Y Plane @ Z=0 view is still active. The area object will now be meshed rather than subdivided as done for the frame objects.

- A. Draw a Selection Box from Left to Right around each of the bottom chords to select all of the points on each of these frame objects.
- B. Click anywhere on the area object to select the deck. Items currently selected are indicated in the lower left corner of the interface.
- C. Click the Edit menu > Mesh Areas command to bring up the Mesh Selected Shells form.

The area object representing the deck was drawn as a single object, but needs to be re-meshed into additional objects so that there will be

connectivity between the deck and the intermediate points along the chord elements. Meshing creates new objects – subdividing does not.

D. Click on the *Mesh using selected Joints on edges* option, and then click **OK**. The model now appears as shown in Figure 22.

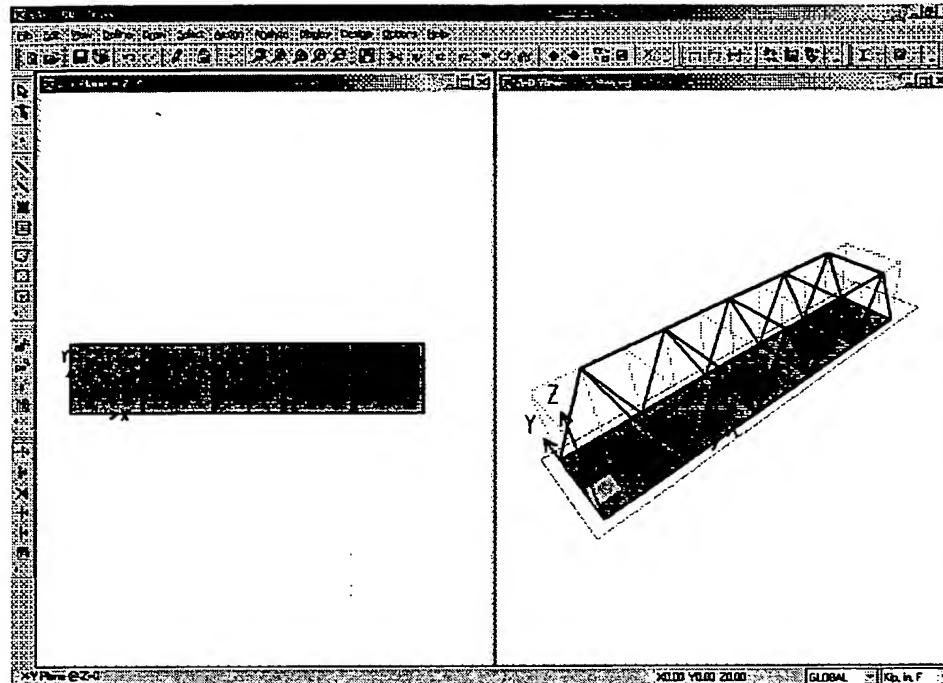


Figure 22
Model after re-mesh
of DECK area object

Step 4 Add Restraints

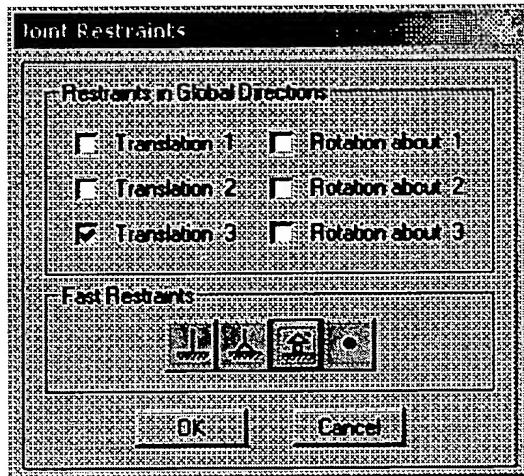
In this step, supports for the truss bridge are defined. Make sure that the X-Y Plane @ Z=0 view is still active, and that the program is in the select mode.

A. Click on the two joints marking the right ends of the two bottom chords.

B. Click on the **Assign** menu > **Joint** > **Restraints** command to bring up the Joint Restraints form as shown in Figure 23.

2

*Figure 23
Joint
Restraints
form*



C. Click on the roller  button to assign restraints in the Translation 3 direction for these two joints. Click **OK** to accept the changes.

D. Click on the two joints marking the left ends of the two bottom chords. The lower left-hand corner of the interface should indicate "2 Points selected".

E. Click on the **Assign** menu > **Joint** > **Restraints** command to bring up the Joint Restraints form.

D. Click on the pinned  button to assign restraints in the Translation 1, 2, & 3 directions for these two joints. Click **OK** to accept the changes.

F. Click the **File** menu > **Save** command, or the **Save**  button, to save your model.

Step 5 Define Load Cases

The loads used in this tutorial consist of dead and live static loads acting in the gravity direction.

For this example, assume that the dead consists of the self weight of the bridge plus an additional 10 pounds per square foot (psf) applied to the concrete deck. The live load is taken to be 100 psf applied to the deck.

- Click the **Define** menu > **Loads** command to bring up the Define Loads form shown in Figure 24. Note there is only a single default load case defined, which is a dead load case with self weight (DEAD).

Note that the self weight multiplier is set to 1 for the default case. This indicates that this load case will automatically include 1.0 times the self weight of all members.

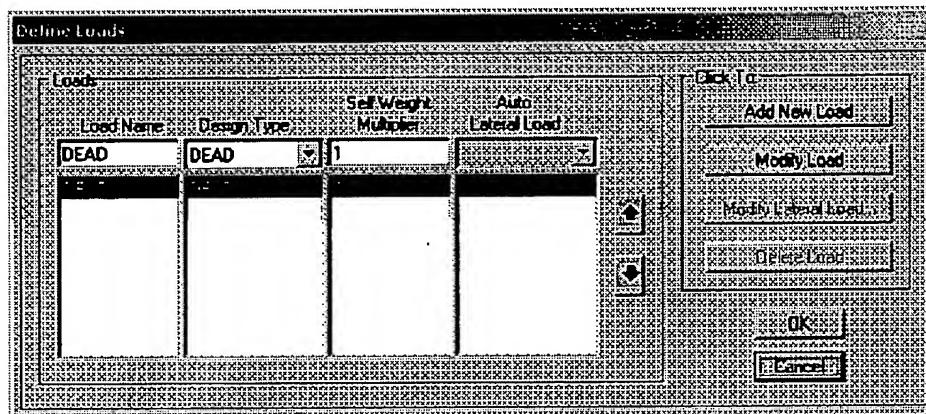


Figure 24
Define Loads form

In SAP2000, both Load Cases and Analysis Cases exist, and they may be different. However, the program automatically creates a corresponding analysis case when a load case is defined, and the analysis cases are available for review at the time the analysis is run.

- Click in the edit box for the Load Name column. Type the name of the new load, **LIVE**. Select a Type of load from the pull down menu;

2

in this case, select *LIVE*. Make sure that the Self Weight Multiplier is set to zero. Click the Add New Load button to add the *LIVE* load to the load list.

The Define Loads form should now appear as shown in Figure 25. Click the **OK** button in that form to accept all of the newly defined static load cases.

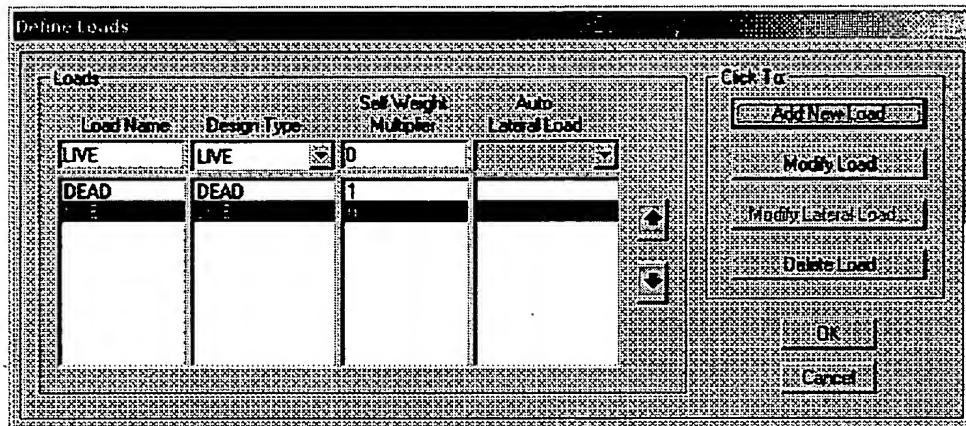


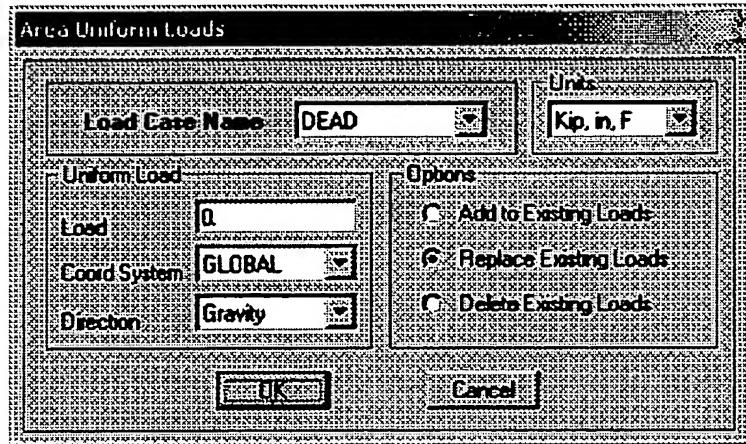
Figure 25
The Define Loads form after
all load cases have been defined

Step 6 Assign Gravity Loads

In this Step, the dead and live gravity loads will be applied to the model. Make sure that the X-Y Plane @ Z=0 view is still active, and that the program is in the select mode.

- A. Draw a Selection Box from Right to Left across the entire deck to select all of the deck objects. The status bar in the lower left-hand corner should show "5 Areas Selected". If you make a mistake in selecting, press the **Clear Selection**  button, and try again.
- B. Click the **Assign menu > Area Loads > Uniform (Shell)** command. This brings up the Area Uniform Loads form. Select *DEAD* from the Load Case Name drop-down box as shown in Figure 26.

Figure 26
Area Uniform
Loads form



1. Select *lb-ft* from the Units drop-down box.
2. Type 10 in the Load edit box in the Uniform Load area.

Again, remember that the *Gravity* Direction is in the negative Global Z direction.

3. Click the **OK** button to accept the dead load.
- C. Draw a Selection Box from Right to Left across the entire deck, or click **Select menu > Get Previous Selection** command, or click the **Get Previous Selection**  button. These actions select all of the deck objects.
- D. Click the **Assign menu > Area Loads > Uniform (Shell)** command. This brings up the Area Uniform Loads form. Select *LIVE* from the Load Case Name drop-down box.
 1. Select *lb-ft* from the Units drop-down box.
 2. Type 100 in the Load edit box in the Uniform Load area.
 3. Click the **OK** button to accept the live load.
- E. Click the **Assign menu > Clear Display of Assigns** command to clear the display of the assigned loads.

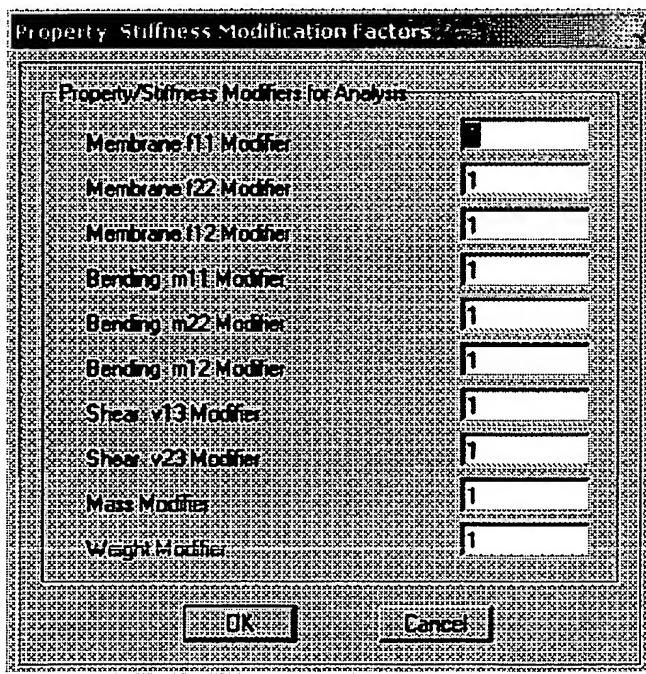
2

Step 7 Assign Area Stiffness Modifiers

In this Step, the membrane properties of the Area objects are modified to prohibit the deck from acting as a flange for the bottom chords of the trusses. Make sure that the X-Y Plane @ Z=0 view is still active, and that the program is in the select mode.

- A. Draw a Selection Box from Right to Left across the entire deck, or click **Select menu > Get Previous Selection** command, or click the **Get Previous Selection**  button. These actions select all of the deck objects.
- B. Click the **Assign menu > Area > Area Stiffness Modifiers** command to bring up the **Property/Stiffness Modification Factors** form shown in Figure 27.

*Figure 27
Property/Stiffness
Modification
Factors form*



1. Type **0** in the Membrane f11 Modifier edit box.

2. Type 0 in the Membrane f22 Modifier edit box.

These actions will prohibit the deck objects from carrying in-plane axial loads.

3. Click **OK** to accept these changes.

C. Click the **Assign** menu > **Clear Display of Assigns** command to clear the display of the stiffness modifiers.

D. Make the 3-D View active by clicking anywhere in the window, and click the **View** menu > **Show Grid** command. This will shut off the grid lines in the 3-D View providing a less cluttered image of the model.

E. Click the **File** menu > **Save** command, or the **Save**  button, to save your model.

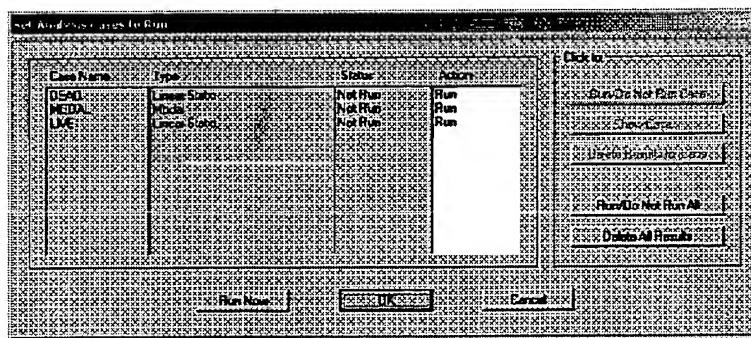
2

Step 8 Run the Analysis

In this Step, the analysis will be run.

A. Click the **Analyze** menu > **Run Analysis** command or the **Run Analysis**  button, to bring up the **Set Analysis Cases to Run** form as shown in Figure 28.

Figure 28
Set
Analysis
Cases
to Run
form



2

Note that the program has automatically defined three different analysis cases: DEAD, MODAL and LIVE based on the load cases defined previously, as well as the assumption that the program may need modal properties for some analysis options, even though no dynamic functions have been defined.

1. Select **MODAL** from the Case Name box.
2. Click the **Run/Do Not Run Case** button to set the action for **MODAL** to *Do Not Run*, as we intend to run only a static analysis.
3. Click the **Run Now** button.

The program will create the analysis model from your object-based SAP2000 model, and will soon display an "Analyzing, Please Wait" window. Data will scroll in this window as the program runs the analysis. This information may be accessed at a later time by going to the **File menu > Show Input/Output Text Files** command and selecting the file with the **.LOG** extension.

- B. When the analysis is finished, the message "ANALYSIS COMPLETE" will display. Click **OK** to close the analysis window. The program automatically displays a deformed shape view of the model, and the model is locked. The model is locked when the **Lock/Unlock Model** button, , appears depressed. Locking the model prevents any changes to the model that would invalidate the analysis results.

Step 9 Graphically Review the Analysis Results

In this Step, the analysis results will be reviewed using graphical representation of the results.

- A. Make sure that the **X-Y Plane @ Z=0** view is active. Then click on the **XZ View**  button to reset the view to an elevation.
- B. Click the **Show Frames/Cables Forces/Stresses** button, , or the **Display menu > Show Forces/Stresses > Frames/Cables** command

to bring up the Member Force Diagram for Frames form shown in Figure 29.

1. Select **DEAD** from the Case/Combo drop-down box.
2. Select the **Axial Force** option.
3. Check the **Fill Diagram**.
4. Click the **OK** button to generate the axial force diagram shown in Figure 30.

Figure 30
Axial force
diagram in
an elevation
view

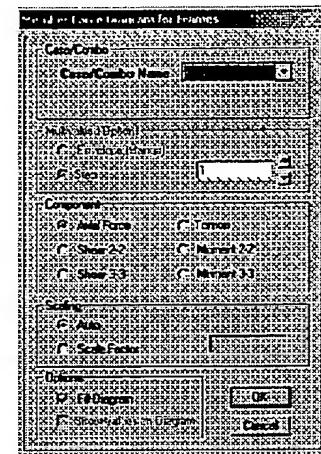


Figure 29
Member Force
Diagram for
Frames form

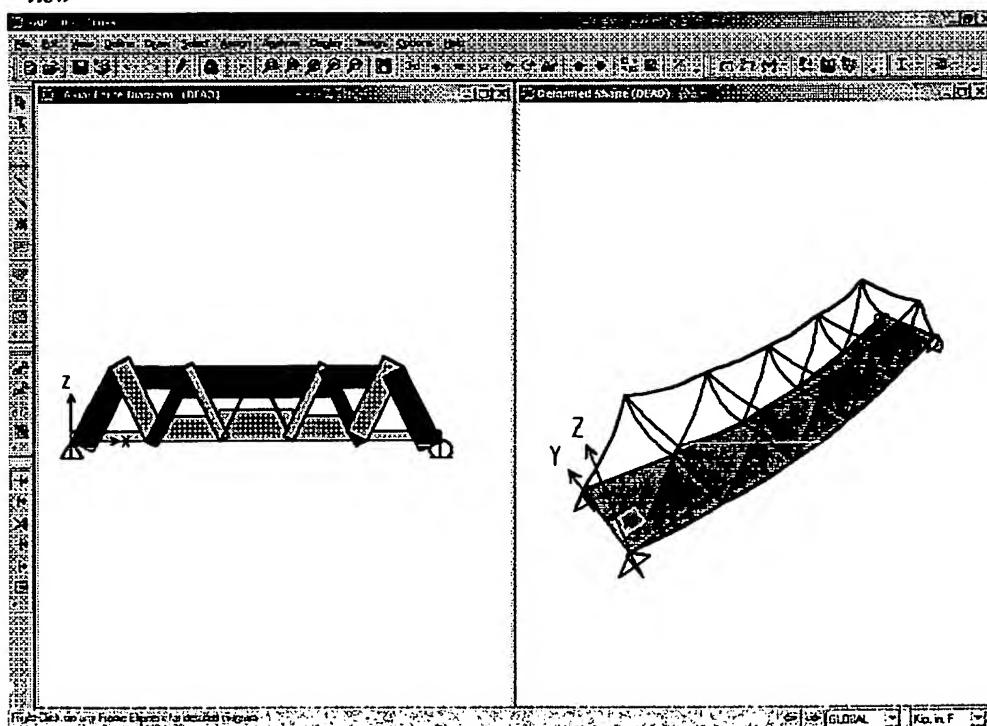
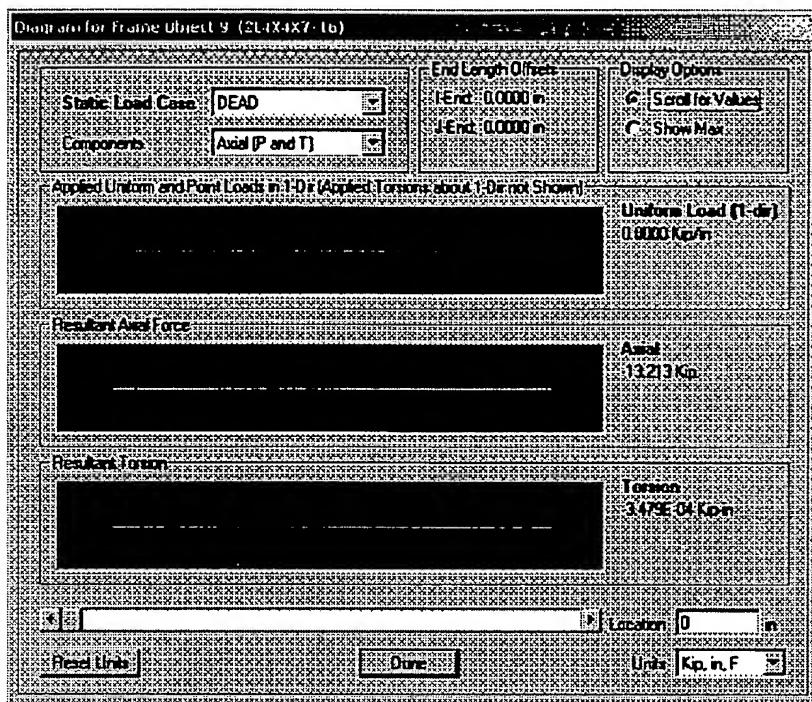


Figure 31
Force details obtained by right-clicking top chord of truss in the elevation view in Figure 29



Note that the program displays the force diagrams for the entire top chord object just as it was drawn, even though the program has automatically subdivided the frame object into smaller elements for analysis.

1. Click the *Scroll for Values* option and a scroll bar appears at the bottom of the form. Drag the scroll bar with your mouse to see values at different locations along the beam.
2. Click the **Done** button to close the form.

D. Make sure that the X-Z View is active and then click the **Display menu** > **Show Deformed Shape** command or the **Show Deformed**

Shape  button, to bring up the Deformed Shape form shown in Figure 32.

1. Select *LIVE* from the Case/Combo Name drop-down box.
2. Check the *Cubic Curve*.
3. Click the **OK** button to generate the deformed shape shown in Figure 33.

Figure 32
Deformed
Shape form

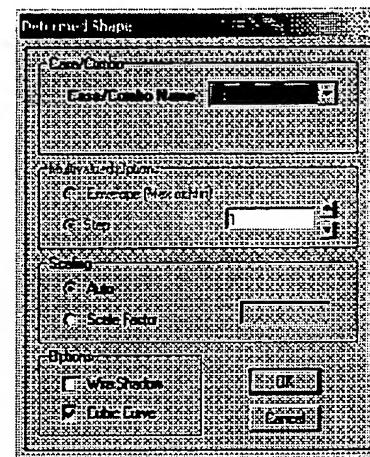
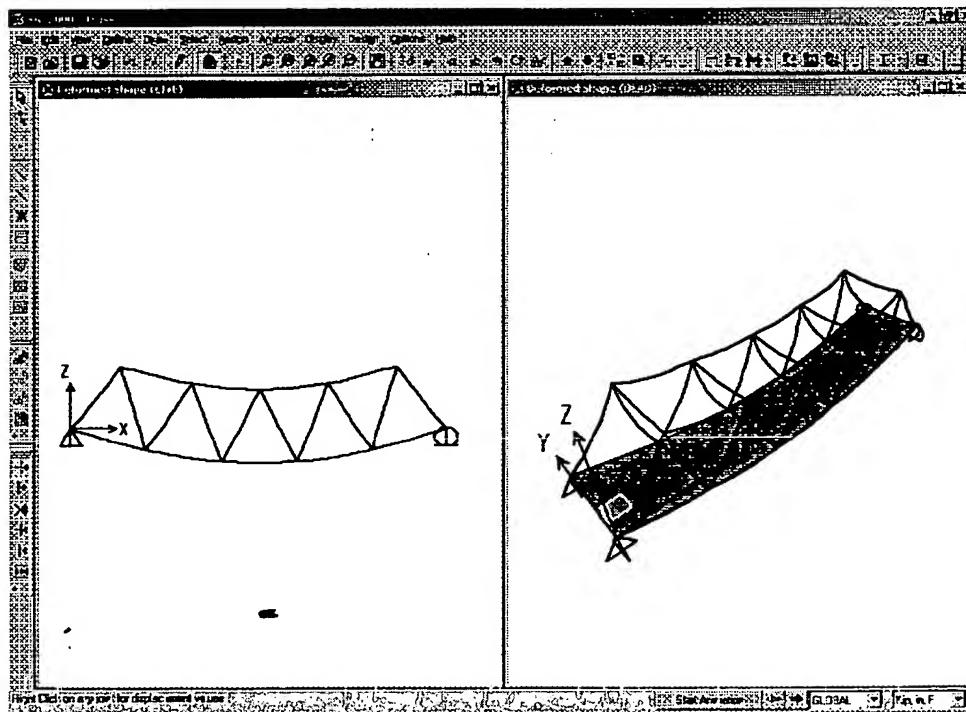


Figure 33
Deformed
Shape
in an elevation
view



E. Right click on the middle joint on the top chord object in Figure 33 to display the Joint Displacements results form shown in Figure 34.

2

Figure 34
Joint Displacements obtained by right-clicking a joint shown in the elevation view in Figure 32

Note that local object axis 3 is in the positive global Z direction.

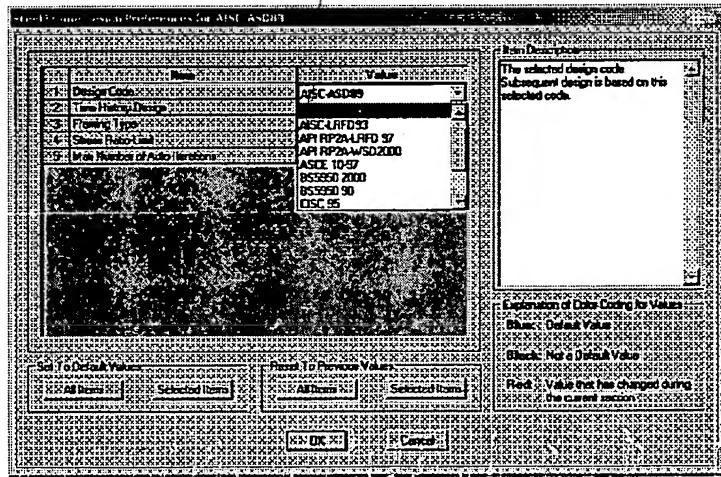
F. Close the Joint Displacements form.

Step 10 Design the Steel Frame Objects

In this Step, the steel frame members of the trusses will be designed. Note that the analysis should be run before completing the following Action Items.

A. Click the Options menu > Preferences > Steel Frame Design command. The Steel Frame Design Preferences form shown in Figure 35 appears.

Figure 35
Steel Frame Design Preferences form



1. Click in the Design Code Values edit box to see the available design codes. Select the *AISC-ASD89* code.
2. Review the information contained in the other edit boxes and then click **OK** to accept the selections.

B. Click the **Design** menu > **Steel Frame Design** > **Start Design/Check of Structure** command or the **Start Steel Design/Check of Structure**  button, to start the steel frame design process. The program designs the steel members, selecting the optimum member size from the TRUSS auto select section list assigned to them when they were drawn.

When the design is complete, the selected sizes are displayed on the model. The model appears as shown in Figure 36.

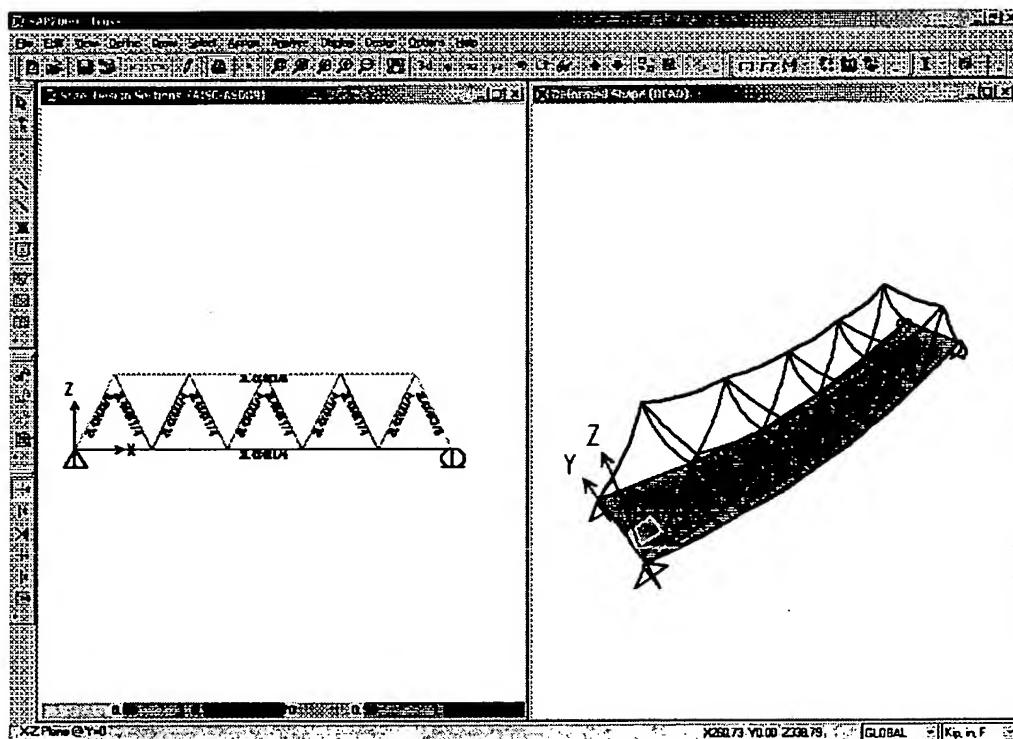
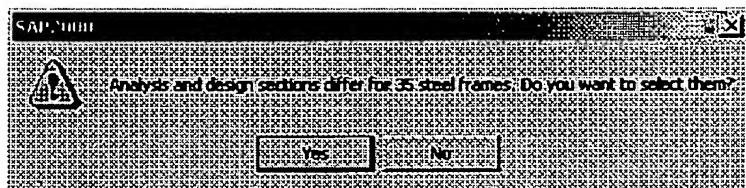


Figure 36
Model after the initial steel frame design

C. Click the Design menu > Steel Frame Design > Verify Analysis vs Design Section command. A message similar to the one in Figure 37 appears. Click the No button to close the form.

2

*Figure 37
Analysis vs Design
Section warning
message*



In the initial analysis (Step 8), the program used the median section by weight from the TRUSS auto select section list. During design (this Step), the program selected different sections than those that were used in the analysis. The message in Figure 37 indicates that the analysis and design sections are different.

The goal is to repeat the analysis and design process until the analysis and design sections are all the same. Note that when the bridge is reanalyzed, SAP2000 will use the current design sections (i.e., those selected in Step 10) as new analysis sections for the next analysis run.

D. Right click on one of the truss top chord member in the X-Z view shown in Figure 36. The Steel Stress Check Information form shown in Figure 38 appears. Note that the reported analysis and design sections are different.

The main body of the form lists the design stress ratios obtained at various stations along the frame object for each design load combination. Note that the program automatically created code-specific design load combinations for this steel frame design.

Also note that the program designed the chord as a single physical member, just as it was drawn as a single object, even though the program has automatically subdivided the frame object into smaller elements for analysis.

2

Steel Stress Check Information							
Frame ID:	9	Analyze Section:	2L6X47/16				
Design Code:	AISC-ASD89	Design Section:	2L4X43/8				
COMBO:	STATION /	HORIZONTAL INTERACTION CHECK /	MAX-SHR /				
ID:	LOC / RATIO	AXL / B-MIN + B-MAX / RATIO	MIN-SHR / RATIO				
DSTL1	432.00	0.254(C) = 0.216 + 0.037 + 0.000	0.003	0.000			
DSTL1	504.00	0.252(C) = 0.216 + 0.036 + 0.000	0.000	0.000			
DSTL1	576.00	0.269(C) = 0.216 + 0.052 + 0.000	0.003	0.000			
DSTL2	0.00	0.551(C) = 0.451 + 0.098 + 0.001	0.004	0.000			
DSTL2	72.00	0.508(C) = 0.451 + 0.056 + 0.000	0.000	0.000			
DSTL2	144.00	0.468(C) = 0.451 + 0.017 + 0.000	0.003	0.000			
DSTL1	144.00	0.765(F) = 0.502 + 0.111 + 0.000	0.001	0.000			

Modify Stress Diagrams Display Details for Selected Item Display Complete Details

Overrides Details User Data

Current System/Program Default

OK Cancel Table Formatted

Figure 38
Steel Stress Check Information form

Click the Details button on the Steel Stress Check Information form. The Steel Stress Check Information AISC-ASD89 form shown in Figure 39 appears. Note that you can print this information using the File menu on the form.

AISC-ASD89 STEEL SECTION CHECK										
Units [In. in. F. in.]										
Combo:	9	Design Sect:	2L4X43/8							
Units:	in. in. F.	Design Type:	Beam							
1 Mid:	246.000	1 End:	246.000	2 End:	246.000	3 End:	246.000	4 End:	246.000	
2 Mid:	144.000	2 End:	144.000	3 End:	144.000	4 End:	144.000	5 End:	144.000	
Length:	576.000	Sect Class:	Non-compact							
BLF:	1.000	Major Axis:	0.000 degrees counterclockwise from local 3							
Area:	5.718	Major:	3.818	Minor:	3.989					
Major:	0.648	Minor:	0.837	Major:	0.900					
Minor:	0.324	Major:	1.229	Minor:	1.229					
Imag:	0.000	Major:	1.639	Minor:	1.639					
									Fy:	36.000
STRESS CHECK FORCES & MOMENTS										
Location:	P	M01	M02	U1	U2	U3	T			
144.000	0.000	0.000	0.000	0.000	0.000	0.000	0.000			
P/M DEMAND/CAPACITY RATIO										
Total:	P	M _{Major}	M _{Minor}	Ratio:	Ratio:	Ratio:	Ratio:	Status:		
Equation:	0.705	0.077	0.000	+	0.100	0.100	0.950	Check:		
(M7-1)								ON		
AXIAL FORCE DESIGN										
Force:	P	F ₁	F ₂	Allowable:	Allowable:	F ₁	F ₂			
Axial:	-61.074	7.046	18.490	27.588	27.588					
MOMENT DESIGN										
M:	F _b	F _b	F _b	F _c	F _c	G _m	H	L	C _b	
Major Moment:	-2.769	0.917	21.000	19.997	19.997	0.059	1.000	0.250	1.000	
Minor Moment:	0.911	0.800	27.000	19.855	19.855	0.059	1.000	0.250		
SHEAR DESIGN										
U:	F _u	F _u	F _u	Stress:	Stress:	Stress:	Stress:			
Imag:	0.000	0.000	0.000	0.000	0.000	0.000	0.000			

Step 10 Design the Steel Frame Objects

2

Click the **X** in the upper right-hand corner of the Steel Stress Check Information AISC-ASD89 form to close it.

Click the **Cancel** Button to close the Steel Stress Check Information form.

- E. To rerun the analysis with the new analysis sections, click the **Analyze** menu > **Run Analysis** command or the **Run Analysis**  button, and then click the **Run Now** button on the **Set Analysis Cases to Run** form.
- F. When the analysis is complete, click the **OK** button to close the analysis window. Click the **Design** menu > **Steel Frame Design** > **Start Design/Check of Structure** command or the **Start Steel Design/Check of Structure**  button, to start the steel frame design process.
- G. When the design is complete, click the **Design** menu > **Steel Frame Design** > **Verify Analysis vs Design Section** command. A message similar to the one in Figure 40 appears.

Figure 40
Analysis vs Design Section message

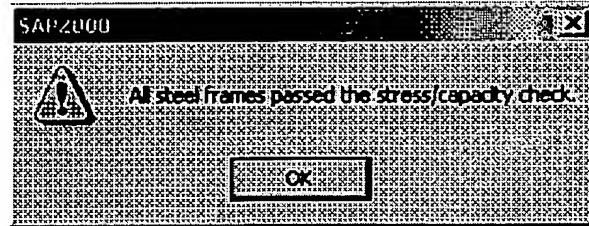


The message in Figure 40 indicates the number of analysis sections that differ from the design sections. Click the **No** button if sections do not match, or the **OK** button if they do match, to close the form.

Repeat Action Items E through G until the message received indicates that all analysis and design sections match. This may take numerous iterations depending upon the complexity of the model.

- H. When the analysis and design sections are the same, click the **Design** menu > **Steel Frame Design** > **Verify all Members Passed** command. A form similar to that shown in Figure 41 should appear indicating that all members passed.

Figure 41
Stress/capacity
check message



Note that members not passing at this stage is an indication of inadequate sections in the auto select list. The program would have used the largest section in the auto select list for both the analysis and design, so the message stating that members do not pass indicates that the auto select section list needs modification. In that case, either add more sections to the auto select sections list or assign larger sections to the members that did not pass and continue the analysis and design iteration.

- I. Click the **File** menu > **Save** command, or the **Save**  button, to save your model.

This introductory tutorial for SAP2000 Version 8 is now complete.

SAP2000®

Integrated
Finite Element Analysis
and
Design of Structures

STEEL DESIGN MANUAL



Computers and Structures, Inc.
Berkeley, California, USA

Version 7.4
Revision May 2000

COPYRIGHT

The computer program SAP2000 and all associated documentation are proprietary and copyrighted products. Worldwide rights of ownership rest with Computers and Structures, Inc. Unlicensed use of the program or reproduction of the documentation in any form, without prior written authorization from Computers and Structures, Inc., is explicitly prohibited.

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

Tel: (510) 845-2177
Fax: (510) 845-4096
E-mail: info@csiberkeley.com
Web: www.csiberkeley.com

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF SAP2000. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THIS PROGRAM IS A VERY PRACTICAL TOOL FOR THE DESIGN/ CHECK OF STEEL STRUCTURES. HOWEVER, THE USER MUST THOROUGHLY READ THE MANUAL AND CLEARLY RECOGNIZE THE ASPECTS OF STEEL DESIGN THAT THE PROGRAM ALGORITHMS DO NOT ADDRESS.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

Table of Contents

CHAPTER I	Introduction	1
Overview	1	
Organization	3	
Recommended Reading	3	
CHAPTER II	Design Algorithms	5
Design Load Combinations	6	
Design and Check Stations	7	
P-Δ Effects	8	
Element Unsupported Lengths	9	
Effective Length Factor (K)	10	
Choice of Input Units	13	
CHAPTER III	Check/Design for AISC-ASD89	15
Design Loading Combinations	18	
Classification of Sections	18	
Calculation of Stresses	22	
Calculation of Allowable Stresses	23	
Allowable Stress in Tension	23	
Allowable Stress in Compression	23	
Flexural Buckling	23	
Flexural-Torsional Buckling	25	
Allowable Stress in Bending	30	
I-sections	30	
Channel sections	33	
T-sections and Double angles	34	
Box Sections and Rectangular Tubes	35	
Pipe Sections	36	
Round Bars	36	

Rectangular and Square Bars	36
Single-Angle Sections	37
General Sections	39
Allowable Stress in Shear	39
Calculation of Stress Ratios	40
Axial and Bending Stresses	41
Shear Stresses	43
CHAPTER IV Check/Design for AISC-LRFD93	45
Design Loading Combinations	48
Classification of Sections	48
Calculation of Factored Forces	52
Calculation of Nominal Strengths	54
Compression Capacity	54
Flexural Buckling	54
Flexural-Torsional Buckling	58
Torsional and Flexural-Torsional Buckling	58
Tension Capacity	60
Nominal Strength in Bending	61
Yielding	61
Lateral-Torsional Buckling	61
Flange Local Buckling	65
Web Local Buckling	69
Shear Capacities	72
Calculation of Capacity Ratios	73
Axial and Bending Stresses	73
Shear Stresses	74
CHAPTER V Check/Design for AASHTO 1997	75
Design Loading Combinations	78
Classification of Sections	79
Calculation of Factored Forces	79
Calculation of Nominal Strengths	82
Compression Capacity	83
Tension Capacity	84
Flexure Capacity	84
Shear Capacities	90
Calculation of Capacity Ratios	91
Axial and Bending Stresses	92
Shear Stresses	92
CHAPTER VI Check/Design for CISC94	93
Design Loading Combinations	96
Classification of Sections	97

Table of Contents

Calculation of Factored Forces	97
Calculation of Factored Strengths	100
Compression Strength	100
Tension Strength	101
Bending Strengths	101
I-shapes and Boxes	102
Rectangular Bar	103
Pipes and Circular Rods	103
Channel Sections	104
T-shapes and double angles	104
Single Angle and General Sections	105
Shear Strengths	105
Calculation of Capacity Ratios	107
Axial and Bending Stresses	107
Shear Stresses	110
CHAPTER VII Check/Design for BS 5950	111
Design Loading Combinations	114
Classification of Sections	115
.	117
Calculation of Factored Forces	117
Calculation of Section Capacities	119
Compression Resistance	119
Tension Capacity	121
Moment Capacity	121
Plastic and Compact Sections	121
Semi-compact Sections	122
Lateral-Torsional Buckling Moment Capacity	122
Shear Capacities	125
Calculation of Capacity Ratios	125
Local Capacity Check	127
Under Axial Tension	127
Under Axial Compression	127
Overall Buckling Check	127
Shear Capacity Check	128
CHAPTER VIII Check/Design for EUROCODE 3	129
Design Loading Combinations	132
Classification of Sections	133
Calculation of Factored Forces	137
Calculation of Section Resistances	138
Tension Capacity	139
Compression Resistance	139
Shear Capacity	141

SAP2000 Steel Design Manual

Moment Resistance	142
Lateral-torsional Buckling.	143
Calculation of Capacity Ratios.	145
Bending, Axial Compression, and Low Shear	145
Bending, Axial Compression, and High Shear	146
Bending, Compression, and Flexural Buckling	146
Bending, Compression, and Lateral-Torsional Buckling	147
Bending, Axial Tension, and Low Shear	148
Bending, Axial Tension, and High Shear	148
Bending, Axial Tension, and Lateral-Torsional Buckling	149
Shear.	149
CHAPTER IX Design Output	151
Overview	151
Graphical Display of Design Output	152
Tabular Display of Design Output	153
Member Specific Information	154
References	157
Index	159

Chapter I

Introduction

Overview

SAP2000 features powerful and completely integrated modules for design of both steel and reinforced concrete structures. The program provides the user with options to create, modify, analyze and design structural models, all from within the same user interface. The program is capable of performing initial member sizing and optimization from within the same interface.

The program provides an interactive environment in which the user can study the stress conditions, make appropriate changes, such as revising member properties, and re-examine the results without the need to re-run the analysis. A single mouse click on an element brings up detailed design information. Members can be grouped together for design purposes. The output in both graphical and tabulated formats can be readily printed.

The program is structured to support a wide variety of the latest national and international design codes for the automated design and check of concrete and steel frame members. The program currently supports the following steel design codes:

- U.S. AISC/ASD (1989),
- U.S. AISC/LRFD (1994),
- U.S. AASHTO LRFD (1997),

- Canadian CAN/CSA-S16.1-94 (1994),
- British BS 5950 (1990), and
- Eurocode 3 (ENV 1993-1-1).

The design is based upon a set of user-specified loading combinations. However, the program provides a set of default load combinations for each design code supported in SAP2000. If the default load combinations are acceptable, no definition of additional load combination is required.

In the design process the program picks the least weight section required for strength for each element to be designed, from a set of user specified sections. Different sets of available sections can be specified for different groups of elements. Also several elements can be grouped to be designed to have the same section.

In the check process the program produces demand/capacity ratios for axial load and biaxial moment interactions and shear. The demand/capacity ratios are based on element stress and allowable stress for allowable stress design, and on factored loads (actions) and factored capacities (resistances) for limit state design.

The checks are made for each user specified (or program defaulted) load combination and at several user controlled stations along the length of the element. Maximum demand/capacity ratios are then reported and/or used for design optimization.

All allowable stress values or design capacity values for axial, bending and shear actions are calculated by the program. Tedious calculations associated with evaluating effective length factors for columns in moment frame type structures are automated in the algorithms.

The presentation of the output is clear and concise. The information is in a form that allows the designer to take appropriate remedial measures if there is member overstress. Backup design information produced by the program is also provided for convenient verification of the results.

Special requirements for seismic design are not implemented in the current version of SAP2000.

English as well as SI and MKS metric units can be used to define the model geometry and to specify design parameters.

Organization

This manual is organized in the following way:

Chapter II outlines various aspects of the steel design procedures of the SAP2000 program. This chapter describes the common terminology of steel design as implemented in SAP2000.

Each of six subsequent chapters gives a detailed description of a specific code of practice as interpreted by and implemented in SAP2000. Each chapter describes the design loading combinations to be considered; allowable stress or capacity calculations for tension, compression, bending, and shear; calculations of demand/capacity ratios; and other special considerations required by the code.

- Chapter III gives a detailed description of the AISC ASD code (AISC 1989) as implemented in SAP2000.
- Chapter IV gives a detailed description of the AISC LRFD code (AISC 1994) as implemented in SAP2000.
- Chapter V gives a detailed description of the AASHTO LRFD steel code (AASHTO 1997) as implemented in SAP2000.
- Chapter VI gives a detailed description of the Canadian code (CISC 1994) as implemented in SAP2000.
- Chapter VII gives a detailed description of the British code BS 5950 (BSI 1990) as implemented in SAP2000.
- Chapter VIII gives a detailed description of the Eurocode 3 (CEN 1992) as implemented in SAP2000.

Chapter IX outlines various aspects of the tabular and graphical output from SAP2000 related to steel design.

Recommended Reading

It is recommended that the user read Chapter II “Design Algorithms” and one of six subsequent chapters corresponding to the code of interest to the user. Finally the user should read “Design Output” in Chapter IX for understanding and interpreting SAP2000 output related to steel design.

A steel design tutorial is presented in the chapter “Steel Design Tutorial” in the *SAP2000 Quick Tutorial* manual. It is recommended that first time users follow through the steps of this tutorial before reading this manual.

Chapter II

Design Algorithms

This chapter outlines various aspects of the steel check and design procedures that are used by the SAP2000 program. The steel design and check may be performed according to one of the following codes of practice.

- American Institute of Steel Construction's "Allowable Stress Design and Plastic Design Specification for Structural Steel Buildings", **AISC-ASD** (AISC 1989).
- American Institute of Steel Construction's "Load and Resistance Factor Design Specification for Structural Steel Buildings", **AISC-LRFD** (AISC 1994).
- American Association of State Highway and Transportation Officials' "AASHTO-LRFD Bridge Design Specifications", **AASHTO-LRFD** (AASHTO 1997).
- Canadian Institute of Steel Construction's "Limit States Design of Steel Structures", **CAN/CSA-S16.1-94** (CISC 1995).
- British Standards Institution's "Structural Use of Steelwork in Building", **BS 5950** (BSI 1990).
- European Committee for Standardization's "Eurocode 3: Design of Steel Structures C Part 1.1: General Rules and Rules for Buildings", **ENV 1993-1-1** (CEN 1992).

Details of the algorithms associated with each of these codes as implemented and interpreted in SAP2000 are described in subsequent chapters. However, this chapter provides a background which is common to all the design codes.

It is assumed that the user has an engineering background in the general area of structural steel design and familiarity with at least one of the above mentioned design codes.

For referring to pertinent sections of the corresponding code, a unique prefix is assigned for each code. For example, all references to the AASHTO-LRFD code carry the prefix of "AASHTO". Similarly,

- References to the AISC-ASD89 code carry the prefix of "ASD"
- References to the AISC-LRFD93 code carry the prefix of "LRFD"
- References to the Canadian code carry the prefix of "CISC"
- References to the British code carry the prefix of "BS"
- References to the Eurocode carry the prefix of "EC3"

Design Load Combinations

The design load combinations are used for determining the various combinations of the load cases for which the structure needs to be designed/checked. The load combination factors to be used vary with the selected design code. The load combination factors are applied to the forces and moments obtained from the associated load cases and the results are then summed to obtain the factored design forces and moments for the load combination.

For multi-valued load combinations involving response spectrum, time history, moving loads and multi-valued combinations (of type enveloping, square-root of the sum of the squares or absolute) where any correspondence between interacting quantities is lost, the program automatically produces multiple sub combinations using maxima/minima permutations of interacting quantities. Separate combinations with negative factors for response spectrum cases are not required because the program automatically takes the minima to be the negative of the maxima for response spectrum cases and the above described permutations generate the required sub combinations.

When a design combination involves only a single multi-valued case of time history or moving load, further options are available. The program has an option to request that time history combinations produce sub combinations for each time step of the time history. Also an option is available to request that moving load combina-

6 Design Load Combinations

tions produce sub combinations using maxima and minima of each design quantity but with corresponding values of interacting quantities.

For normal loading conditions involving static dead load, live load, wind load, and earthquake load, and/or dynamic response spectrum earthquake load, the program has built-in default loading combinations for each design code. These are based on the code recommendations and are documented for each code in the corresponding chapters.

For other loading conditions involving moving load, time history, pattern live loads, separate consideration of roof live load, snow load, etc., the user must define design loading combinations either in lieu of or in addition to the default design loading combinations.

The default load combinations assume all static load cases declared as dead load to be additive. Similarly, all cases declared as live load are assumed additive. However, each static load case declared as wind or earthquake, or response spectrum cases, is assumed to be non additive with each other and produces multiple lateral load combinations. Also wind and static earthquake cases produce separate loading combinations with the sense (positive or negative) reversed. If these conditions are not correct, the user must provide the appropriate design combinations.

The default load combinations are included in design if the user requests them to be included or if no other user defined combination is available for concrete design. If any default combination is included in design, then all default combinations will automatically be updated by the program any time the user changes to a different design code or if static or response spectrum load cases are modified.

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

The user is cautioned that if moving load or time history results are not requested to be recovered in the analysis for some or all the frame members, then the effects of these loads will be assumed to be zero in any combination that includes them.

Design and Check Stations

For each load combination, each element is designed or checked at a number of locations along the length of the element. The locations are based on equally spaced segments along the clear length of the element. The number of segments in an element is requested by the user before the analysis is made. The user can refine the design along the length of an element by requesting more segments.

The axial-flexure interaction ratios as well as shear stress ratios are calculated for each station along the length of the member for each load combination. The actual member stress components and corresponding allowable stresses are calculated. Then, the stress ratios are evaluated according to the code. The controlling compression and/or tension stress ratio is then obtained, along with the corresponding identification of the station, load combination, and code-equation. A stress ratio greater than 1.0 indicates an overstress or exceeding a limit state.

P- Δ Effects

The SAP2000 design algorithms require that the analysis results include the P- Δ effects. The P- Δ effects are considered differently for "braced" or "nonsway" and "unbraced" or "sway" components of moments in frames. For the braced moments in frames, the effect of P- Δ is limited to "individual member stability". For unbraced components, "lateral drift effects" should be considered in addition to individual member stability effect. In SAP2000, it is assumed that "braced" or "nonsway" moments are contributed from the "dead" or "live" loads. Whereas, "unbraced" or "sway" moments are contributed from all other types of loads.

For the individual member stability effects, the moments are magnified with moment magnification factors as in the AISC-LRFD and AASHTO-LRFD codes or are considered directly in the design equations as in the Canadian, British, and European codes. No moment magnification is applied to the AISC-ASD code.

For lateral drift effects of unbraced or sway frames, SAP2000 assumes that the amplification is already included in the results because P- Δ effects are considered for all but AISC-ASD code.

The users of SAP2000 should be aware that the default analysis option in SAP2000 is turned OFF for P- Δ effect. The default number of iterations for P- Δ analysis is 1. **The user should turn the P- Δ analysis ON and set the maximum number of iterations for the analysis.** No P- Δ analysis is required for the AISC-ASD code. For further reference, the user is referred to *SAP2000 Analysis Reference Manual* (CSI 1997).

The user is also cautioned that SAP2000 currently considers P- Δ effects due to axial loads in frame members only. Forces in other types of elements do not contribute to this effect. If significant forces are present in other types of elements, for example, large axial loads in shear walls modeled as shell elements, then the additional forces computed for P- Δ will be inaccurate.

Element Unsupported Lengths

To account for column slenderness effects, the column unsupported lengths are required. The two unsupported lengths are l_{33} and l_{22} . See Figure II-1. These are the lengths between support points of the element in the corresponding directions. The length l_{33} corresponds to instability about the 3-3 axis (major axis), and l_{22} corresponds to instability about the 2-2 axis (minor axis). The length l_{22} is also used for lateral-torsional buckling caused by major direction bending (i.e., about the 3-3 axis). See Figure II-2 for correspondence between the SAP2000 axes and the axes in the design codes.

Normally, the unsupported element length is equal to the length of the element, i.e., the distance between END-I and END-J of the element. See Figure II-1. The program, however, allows users to assign several elements to be treated as a single member for design. This can be done differently for major and minor bending. Therefore, extraneous joints, as shown in Figure II-3, that affect the unsupported length of an element are automatically taken into consideration.

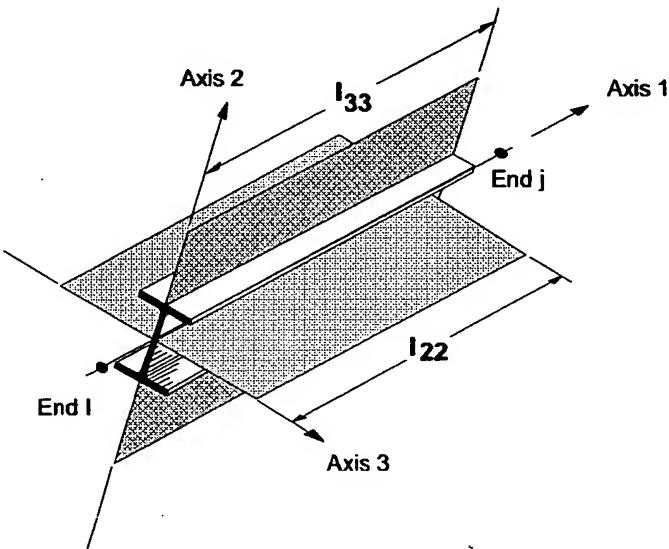


Figure II-1
Major and Minor Axes of Bending

In determining the values for l_{22} and l_{33} of the elements, the program recognizes various aspects of the structure that have an effect on these lengths, such as member connectivity, diaphragm constraints and support points. The program automatically locates the element support points and evaluates the corresponding unsupported element length.

Therefore, the unsupported length of a column may actually be evaluated as being greater than the corresponding element length. If the beam frames into only one direction of the column, the beam is assumed to give lateral support only in that direction. The user has options to specify the unsupported lengths of the elements on an element-by-element basis.

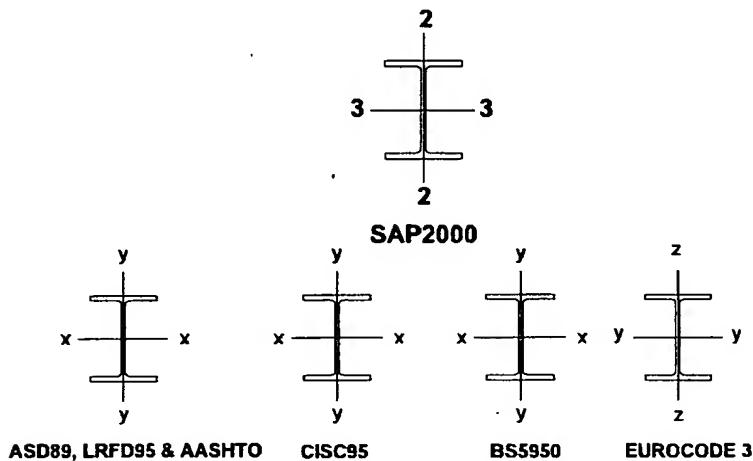


Figure II-2
Correspondence between SAP2000 Axes and Code Axes

Effective Length Factor (K)

The column K -factor algorithm has been developed for building-type structures, where the columns are vertical and the beams are horizontal, and the behavior is basically that of a moment-resisting nature for which the K -factor calculation is relatively complex. For the purpose of calculating K -factors, the elements are identified as columns, beams and braces. All elements parallel to the Z-axis are classified as columns. All elements parallel to the X-Y plane are classified as beams. The rest are braces.

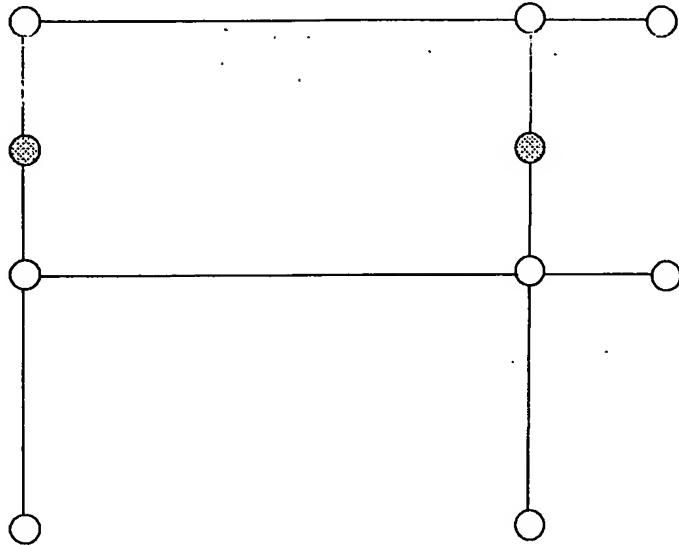


Figure II-3
Unsupported Lengths are Affected by Intermediate Nodal Points

The beams and braces are assigned K -factors of unity. In the calculation of the K -factors for a column element, the program first makes the following four stiffness summations for each joint in the structural model:

$$\begin{aligned} S_{cx} &= \sum \left(\frac{E_c I_c}{L_c} \right)_x & S_{bx} &= \sum \left(\frac{E_b I_b}{L_b} \right)_x \\ S_{cy} &= \sum \left(\frac{E_c I_c}{L_c} \right)_y & S_{by} &= \sum \left(\frac{E_b I_b}{L_b} \right)_y \end{aligned}$$

where the x and y subscripts correspond to the global X and Y directions and the c and b subscripts refer to column and beam. The local 2-2 and 3-3 terms EI_{22}/l_{22} and EI_{33}/l_{33} are rotated to give components along the global X and Y directions to form the $(EI/I)_x$ and $(EI/I)_y$ values. Then for each column, the joint summations at END-I and the END-J of the member are transformed back to the column local 1-2-3 coordinate system and the G -values for END-I and the END-J of the member are calculated about the 2-2 and 3-3 directions as follows:

$$G^I_{22} = \frac{S^I_{c22}}{S^I_{b22}} \quad G^J_{22} = \frac{S^J_{c22}}{S^J_{b22}}$$

$$G^I_{33} = \frac{S^I_{c33}}{S^I_{b33}} \quad G^J_{33} = \frac{S^J_{c33}}{S^J_{b33}}$$

If a rotational release exists at a particular end (and direction) of an element, the corresponding value is set to 10.0. If all degrees of freedom for a particular joint are deleted, the G -values for all members connecting to that joint will be set to 1.0 for the end of the member connecting to that joint. Finally, if G^I and G^J are known for a particular direction, the column K -factor for the corresponding direction is calculated by solving the following relationship for α :

$$\frac{\alpha^2 G^I G^J - 36}{6(G^I + G^J)} = \frac{\alpha}{\tan \alpha}$$

from which $K = \pi / \alpha$. This relationship is the mathematical formulation for the evaluation of K factors for moment-resisting frames assuming sidesway to be uninhibited. For other structures, such as braced frame structures, trusses, space frames, transmission towers, etc., the K -factors for all members are usually unity and should be set so by the user. The following are some important aspects associated with the column K -factor algorithm:

- An element that has a pin at the joint under consideration will not enter the stiffness summations calculated above. An element that has a pin at the far end from the joint under consideration will contribute only 50% of the calculated EI value. Also, beam elements that have no column member at the far end from the joint under consideration, such as cantilevers, will not enter the stiffness summation.
- If there are no beams framing into a particular direction of a column element, the associated G -value will be infinity. If the G -value at any one end of a column for a particular direction is infinity, the K -factor corresponding to that direction is set equal to unity.
- If rotational releases exist at both ends of an element for a particular direction, the corresponding K -factor is set to unity.
- The automated K -factor calculation procedure can occasionally generate artificially high K -factors, specifically under circumstances involving skewed beams, fixed support conditions, and under other conditions where the program may have difficulty recognizing that the members are laterally supported and K -factors of unity are to be used.

- All K -factors produced by the program can be overwritten by the user. These values should be reviewed and any unacceptable values should be replaced.

Choice of Input Units

English as well as SI and MKS metric units can be used for input. But the codes are based on a specific system of units. All equations and descriptions presented in the subsequent chapters correspond to that specific system of units unless otherwise noted. For example, AISC-ASD code is published in kip-inch-second units. By default, all equations and descriptions presented in the chapter "Check/Design for AISC-ASD89" correspond to kip-inch-second units. However, any system of units can be used to define and design the structure in SAP2000.

Chapter III

Check/Design for AISC-ASD89

This chapter describes the details of the structural steel design and stress check algorithms that are used by SAP2000 when the user selects the AISC-ASD89 design code (AISC 1989). Various notations used in this chapter are described in Table III-1.

For referring to pertinent sections and equations of the original ASD code, a unique prefix "ASD" is assigned. However, all references to the "Specifications for Allowable Stress Design of Single-Angle Members" carry the prefix of "ASD SAM".

The design is based on user-specified loading combinations. But the program provides a set of default load combinations that should satisfy requirements for the design of most building type structures.

In the evaluation of the axial force/biaxial moment capacity ratios at a station along the length of the member, first the actual member force/moment components and the corresponding capacities are calculated for each load combination. Then the capacity ratios are evaluated at each station under the influence of all load combinations using the corresponding equations that are defined in this chapter. The controlling capacity ratio is then obtained. A capacity ratio greater than 1.0 indicates overstress. Similarly, a shear capacity ratio is also calculated separately.

A	= Cross-sectional area, in ²
A_e	= Effective cross-sectional area for slender sections, in ²
A_f	= Area of flange, in ²
A_g	= Gross cross-sectional area, in ²
A_{v2}, A_{v3}	= Major and minor shear areas, in ²
A_w	= Web shear area, dt_w , in ²
C_b	= Bending Coefficient
C_m	= Moment Coefficient
C_w	= Warping constant, in ⁶
D	= Outside diameter of pipes, in
E	= Modulus of elasticity, ksi
F_a	= Allowable axial stress, ksi
F_b	= Allowable bending stress, ksi
F_{b33}, F_{b22}	= Allowable major and minor bending stresses, ksi
F_c	= Critical compressive stress, ksi
F'_{33}	= $\frac{12 \pi^2 E}{23(K_{33}l_{33}/r_{33})^2}$
F'_{22}	= $\frac{12 \pi^2 E}{23(K_{22}l_{22}/r_{22})^2}$
F_v	= Allowable shear stress, ksi
F_y	= Yield stress of material, ksi
K	= Effective length factor
K_{33}, K_{22}	= Effective length K -factors in the major and minor directions
M_{33}, M_{22}	= Major and minor bending moments in member, kip-in
M_{ab}	= Lateral-torsional moment for angle sections, kip-in
P	= Axial force in member, kips
P_e	= Euler buckling load, kips
Q	= Reduction factor for slender section, = $Q_a Q_s$
Q_a	= Reduction factor for stiffened slender elements
Q_s	= Reduction factor for unstiffened slender elements
S	= Section modulus, in ³
S_{33}, S_{22}	= Major and minor section moduli, in ³

Table III-1
AISC-ASD Notations

$S_{eff,33}, S_{eff,22}$	Effective major and minor section moduli for slender sections, in ³
S_c	Section modulus for compression in an angle section, in ³
V_2, V_3	Shear forces in major and minor directions, kips
b	Nominal dimension of plate in a section, in longer leg of angle sections, $b_f - 2t_w$ for welded and $b_f - 3t_w$ for rolled box sections, etc.
b_e	Effective width of flange, in
b_f	Flange width, in
d	Overall depth of member, in
f_a	Axial stress either in compression or in tension, ksi
f_b	Normal stress in bending, ksi
f_{b33}, f_{b22}	Normal stress in major and minor direction bending, ksi
f_c	Shear stress, ksi
f_{s2}, f_{s3}	Shear stress in major and minor direction bending, ksi
h	Clear distance between flanges for I shaped sections ($d - 2t_f$), in
h_e	Effective distance between flanges less fillets, in
k	Distance from outer face of flange to web toe of fillet, in
k_c	Parameter used for classification of sections, $\frac{4.05}{[h/t_w]^{0.46}} \text{ if } h/t_w > 70, \\ 1 \text{ if } h/t_w \leq 70.$
l_{33}, l_{22}	Major and minor direction unbraced member lengths, in
l_c	Critical length, in
r	Radius of gyration, in
r_{33}, r_{22}	Radii of gyration in the major and minor directions, in
r_z	Minimum Radius of gyration for angles, in
t	Thickness of a plate in I, box, channel, angle, and T sections, in
t_f	Flange thickness, in
t_w	Web thickness, in
β_w	Special section property for angles, in

Table III-1
AISC-ASD Notations (cont.)

English as well as SI and MKS metric units can be used for input. But the code is based on Kip-Inch-Second units. For simplicity, all equations and descriptions presented in this chapter correspond to **Kip-Inch-Second** units unless otherwise noted.

Design Loading Combinations

The design load combinations are the various combinations of the load cases for which the structure needs to be checked. For the AISC-ASD89 code, if a structure is subjected to dead load (DL), live load (LL), wind load (WL), and earthquake induced load (EL), and considering that wind and earthquake forces are reversible, then the following load combinations may have to be defined (ASD A4):

DL	(ASD A4.1)
DL + LL	(ASD A4.1)
DL \pm WL	(ASD A4.1)
DL + LL \pm WL	(ASD A4.1)
DL \pm EL	(ASD A4.1)
DL + LL \pm EL	(ASD A4.1)

These are also the default design load combinations in SAP2000 whenever the AISC-ASD89 code is used. The user should use other appropriate loading combinations if roof live load is separately treated, if other types of loads are present, or if pattern live loads are to be considered.

When designing for combinations involving earthquake and wind loads, allowable stresses are increased by a factor of 4/3 of the regular allowable value (ASD A5.2).

Live load reduction factors can be applied to the member forces of the live load case on an element-by-element basis to reduce the contribution of the live load to the factored loading.

Classification of Sections

The allowable stresses for axial compression and flexure are dependent upon the classification of sections as either Compact, Noncompact, Slender, or Too Slender. SAP2000 classifies the individual members according to the limiting width/thickness ratios given in Table III-2 (ASD B5.1, F3.1, F5, G1, A-B5-2). The definition of the section properties required in this table is given in Figure III-1 and Table III-1.

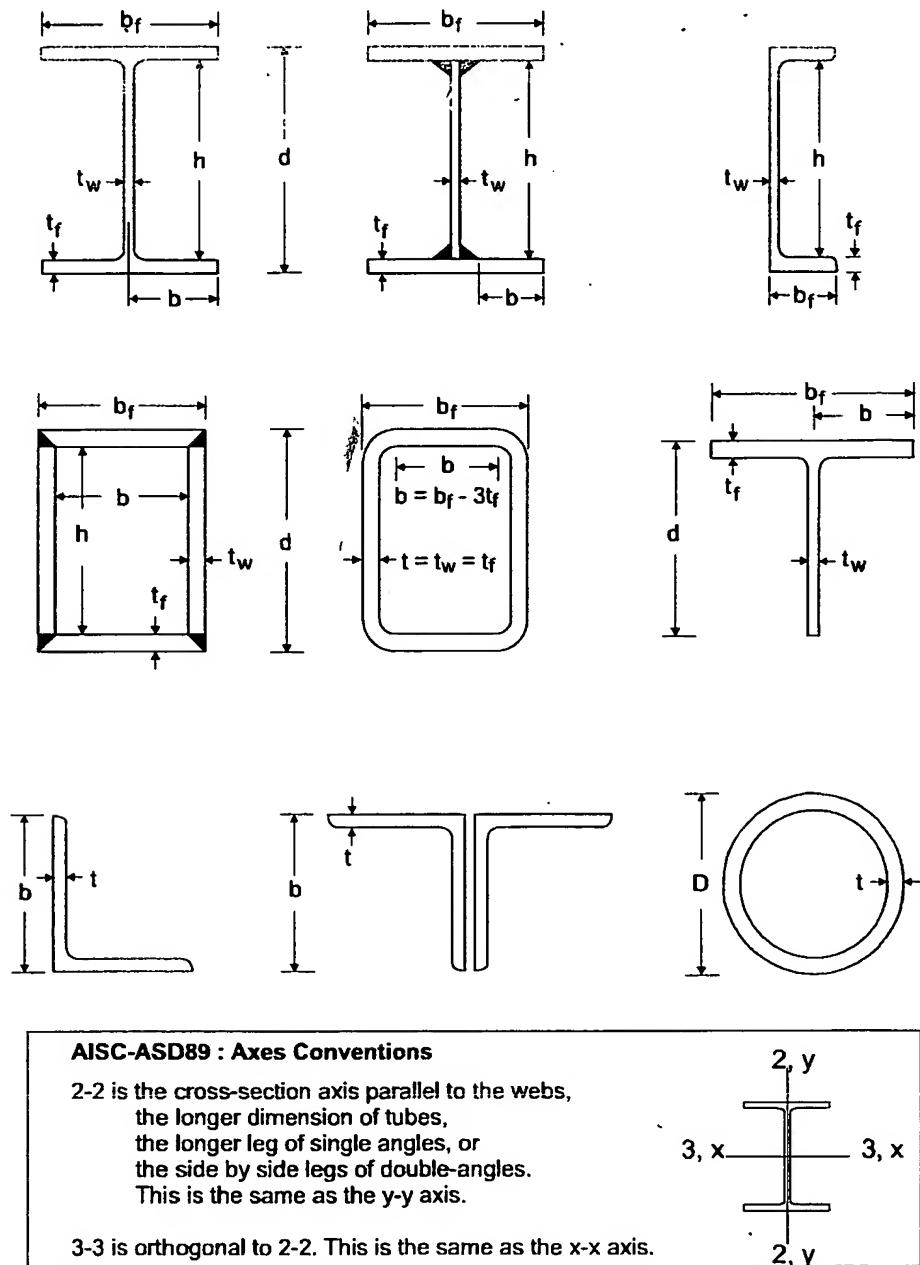


Figure III-1
AISC-ASD Definition of Geometric Properties

Section Description	Ratio Checked	Compact Section	Noncompact Section	Slender Section
I-SHAPE	$b_f / 2t_f$ (rolled)	$\leq 65 / \sqrt{F_y}$	$\leq 95 / \sqrt{F_y}$	No limit
	$b_f / 2t_f$ (welded)	$\leq 65 / \sqrt{F_y}$	$\leq 95 / \sqrt{F_y / k_c}$	No limit
	d / t_w	For $f_a / F_c \leq 0.16$ $\leq \frac{640}{\sqrt{F_y}} (1 - 3.74 \frac{f_a}{F_y})$, For $f_a / F_c > 0.16$ $\leq 257 / \sqrt{F_y}$.	No limit	No limit
	h / t_w	No limit	If compression only, $\leq 253 / \sqrt{F_y}$ otherwise $\leq 760 / \sqrt{F_b}$	$\leq \frac{14000}{\sqrt{F_y (F_y + 16.5)}}$ ≤ 260
BOX	b / t_f	$\leq 190 / \sqrt{F_y}$	$\leq 238 / \sqrt{F_y}$	No limit
	d / t_w	As for I-shapes	No limit	No limit
	h / t_w	No limit	As for I-shapes	As for I-shapes
	Other	$t_w \geq t_f / 2, d_w \leq 6b_f$	None	None
CHANNEL	b / t_f	As for I-shapes	As for I-shapes	No limit
	d / t_w	As for I-shapes	No limit	No limit
	h / t_w	No limit	As for I-shapes	As for I-shapes
	Other	No limit	No limit	If welded $b_f / d_w \leq 0.25$, $t_f / t_w \leq 3.0$ If rolled $b_f / d_w \leq 0.5$, $t_f / t_w \leq 2.0$

Table III-2
*Limiting Width-Thickness Ratios for
 Classification of Sections Based on AISC-ASD*

Chapter IX

Design Output

Overview

SAP2000 creates design output in three different major formats: graphical display, tabular output, and member specific detailed design information.

The graphical display of steel design output includes input and output design information. Input design information includes design section labels, *K*-factors, live load reduction factors, and other design parameters. The output design information includes axial and bending interaction ratios and shear stress ratios. All graphical output can be printed.

The tabular output can be saved in a file or printed. The tabular output includes most of the information which can be displayed. This is generated for added convenience to the designer.

The member-specific detailed design information shows details of the calculation from the designer's point of view. It shows the design section dimensions, material properties, design and allowable stresses or factored and nominal strengths, and some intermediate results for all the load combinations at all the design sections of a specific frame member.

In the following sections, some of the typical graphical display, tabular output, and member-specific detailed design information are described. Some of the design information is specific to the chosen steel design codes which are available in the program and is only described where required. The AISC-ASD89 design code is described in the latter part of this chapter. For all other codes, the design outputs are similar.

Graphical Display of Design Output

The graphical output can be produced either as color screen display or in gray-scaled printed form. Moreover, the active screen display can be sent directly to the printer. The graphical display of design output includes input and output design information.

Input design information, for the AISC-ASD89 code, includes

- Design section labels,
- K -factors for major and minor direction of buckling,
- Unbraced Length Ratios,
- C_m -factors,
- C_b -factors,
- Live Load Reduction Factors,
- δ_s -factors,
- δ_b -factors,
- design type,
- allowable stresses in axial, bending, and shear.

The output design information which can be displayed is

- Color coded P-M interaction ratios with or without values, and
- Color coded shear stress ratios.

The graphical displays can be accessed from the **Design** menu. For example, the color coded P-M interaction ratios with values can be displayed by selecting the **Display Design Info...** from the **Design** menu. This will pop up a dialog box called **Display Design Results**. Then the user should switch on the **Design Output** option button (default) and select **P-M Ratios Colors & Values** in the drop-down box. Then clicking the **OK** button will show the interaction ratios in the active window.

The graphics can be displayed in either 3D or 2D mode. The SAP2000 standard view transformations are available for all steel design input and output displays. For switching between 3D or 2D view of graphical displays, there are several buttons on the main toolbar. Alternatively, the view can be set by choosing **Set 3D View...** from the **View** menu.

The graphical display in an active window can be printed in gray scaled black and white from the SAP2000 program. To send the graphical output directly to the printer, click on the **Print Graphics** button in the **File** menu. A screen capture of the active window can also be made by following the standard procedure provided by the Windows operating system.

Tabular Display of Design Output

The tabular design output can be sent directly either to a printer or to a file. The printed form of tabular output is the same as that produced for the file output with the exception that for the printed output font size is adjusted.

The tabular design output includes input and output design information which depends on the design code of choice. For the AISC-ASD89 code, the tabular output includes the following. All tables have formal headings and are self-explanatory, so further description of these tables is not given.

Input design information includes the following:

- Load Combination Multipliers
 - Combination name,
 - Load types, and
 - Load factors.
- Steel Stress Check Element Information (code dependent)
 - Frame ID,
 - Design Section ID,
 - K -factors for major and minor direction of buckling,
 - Unbraced Length Ratios,
 - C_m -factors,
 - C_b -factors, and
 - Live Load Reduction Factors.

- Steel Moment Magnification Factors (code dependent)
 - Frame ID,
 - Section ID,
 - Framing Type,
 - δ_b -factors, and
 - δ_s -factors.

The output design information includes the following:

- Steel Stress Check Output (code dependent)
 - Frame ID,
 - Section location,
 - Controlling load combination ID for P-M interaction,
 - Tension or compression indication,
 - Axial and bending interaction ratio,
 - Controlling load combination ID for major and minor shear forces, and
 - Shear stress ratios.

The tabular output can be accessed by selecting **Print Design Tables...** from the **File** menu. This will pop up a dialog box. Then the user can specify the design quantities for which the results are to be tabulated. By default, the output will be sent to the printer. If the user wants the output stream to be redirected to a file, he/she can check the **Print to File** box. This will provide a default filename. The default filename can be edited. Alternatively, a file list can be obtained by clicking the **File Name** button to chose a file from. Then clicking the **OK** button will direct the tabular output to the requested stream — the file or the printer.

Member Specific Information

The member specific design information shows the details of the calculation from the designer's point of view. It provides an access to the geometry and material data, other input data, design section dimensions, design and allowable stresses, reinforcement details, and some of the intermediate results for a member. The design detail information can be displayed for a specific load combination and for a specific station of a frame member.

The detailed design information can be accessed by right clicking on the desired frame member. This will pop up a dialog box called **Steel Stress Check Information** which includes the following tabulated information for the specific member

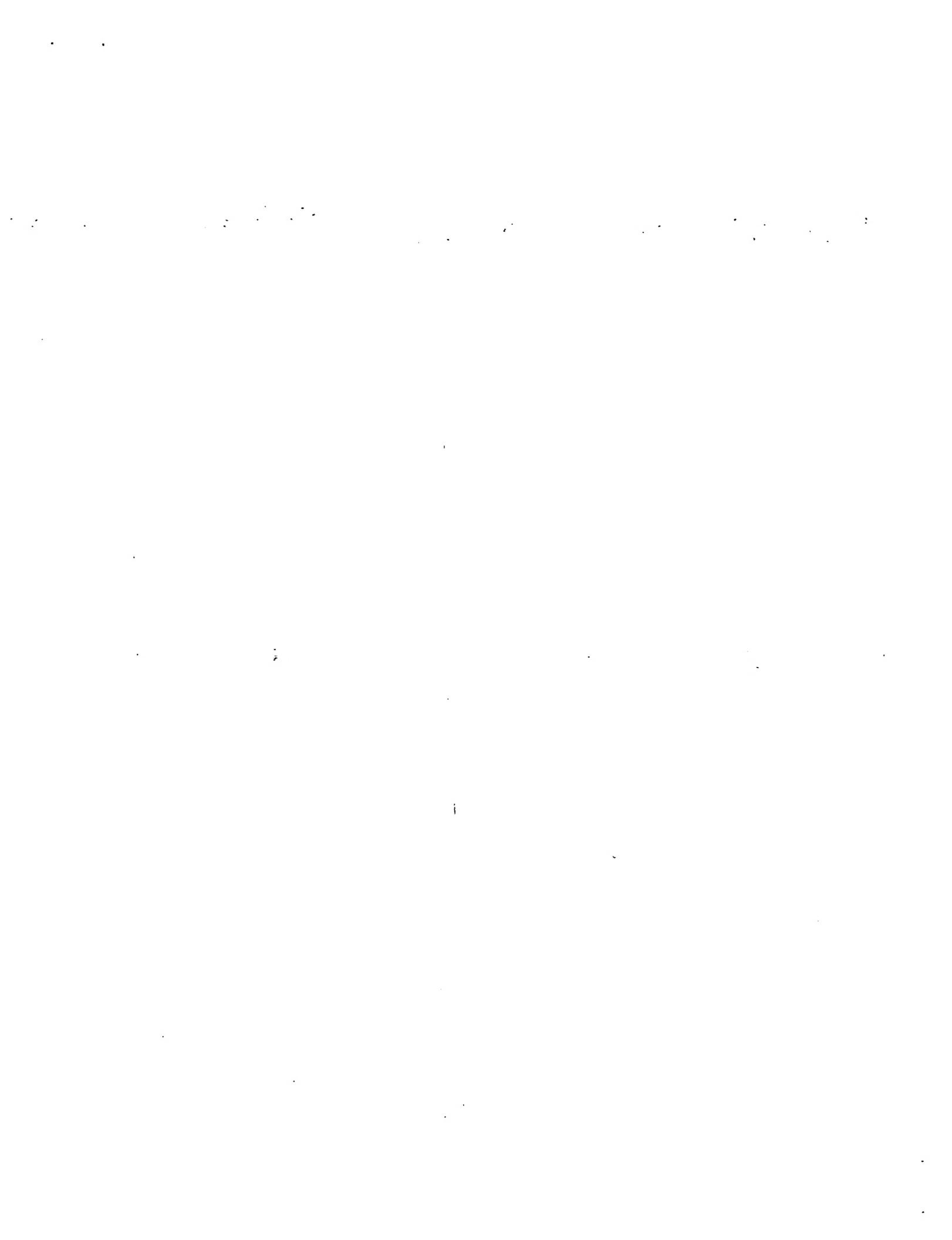
- Frame ID,
- Section ID,
- Load combination ID,
- Station location,
- Axial and bending interaction ratio, and
- Shear stress ratio along two axes.

Additional information can be accessed by clicking on the **ReDesign** and **Details** buttons in the dialog box. Additional information that is available by clicking on the **ReDesign** button is as follows:

- Design Factors (code dependent)
 - Effective length factors, K , for major and minor direction of buckling,
 - Unbraced Length Ratios,
 - C_m -factors,
 - C_b -factors,
 - Live Load Reduction Factors,
 - δ_s -factors, and
 - δ_b -factors.
- Element Section ID
- Element Framing Type
- Overwriting allowable stresses

Additional information that is available by clicking on the **Details** button is given below.

- Frame, Section, Station, and Load Combination IDs,
- Section geometric information and graphical representation,
- Material properties of steel,
- Moment factors,
- Design and allowable stresses for axial force and biaxial moments, and
- Design and allowable stresses for shear.



References

AASHTO, 1997

AASHTO LRFD Bridge Design Specifications — U.S. Units, 1997 Interim Edition, American Association of State Highway and Transportation Officials, 1997.

AISC, 1989

Manual of Steel Construction, Allowable Stress Design, 9th Edition, American Institute of Steel Construction, Chicago, Ill, 1989.

AISC, 1994

Manual of Steel Construction, Load & Resistance Factor Design, 2nd Edition, American Institute of Steel Construction, Chicago, Ill, 1994.

BSI, 1990

Structural Use of Steelwork in Building, Part 1, Code of Practice for Design in Simple and Continuous Construction: Hot Rolled Sections, BS 5950 : Part 1 : 1990, British Standards Institution, London, UK, 1990.

CEN, 1992

Design of Steel Structures, Part 1.1 : General Rules and Rules for Buildings, ENV 1993-1-1 : 1992, European Committee for Standardization, Brussels, Belgium, 1992.

CISC, 1995

Handbook of Steel Construction, CAN/CSA-S16.1-94, 6th Edition, Canadian Institute of Steel Construction, Willowdale, Ontario, Canada, 1995.

CSI, 1998a

SAP2000 Getting Started, Computers and Structures, Inc., Berkeley, California, 1998.

CSI, 1998b

SAP2000 Quick Tutorial, Computers and Structures, Inc., Berkeley, California, 1998.

CSI, 1997

SAP2000 Analysis Reference, Vols. I and II, Computers and Structures, Inc., Berkeley, California, 1997.

ICBO, 1997

Uniform Building Code, 1997, International Conference of Building Officials, Whittier, California, 1997.

D. W. White and J. F. Hajjar, 1991

“Application of Second-Order Elastic Analysis in LRFD: Research to Practice,” *Engineering Journal, American Institute of Steel Construction, Inc.*, Vol. 28, No. 4, 1991.

Index

Bending strength
 AASHTO, 84
 ASD (allowable), 30
 BS, 121
 CISC, 101
 Eurocode, 142
 LRFD, 61

Braced frames, 8
 AASHTO, 79
 BS, 119
 CISC, 97
 Eurocode, 137
 LRFD, 52

Capacity ratio, 2, 8
 AASHTO, 75, 91
 ASD, 15, 40
 BS, 111, 125
 CISC, 93, 107
 Eurocode, 129, 145
 LRFD, 45, 73

Check stations, 7

Classification of sections
 AASHTO, 79
 ASD, 18
 BS, 115
 CISC, 97

Eurocode, 133
LRFD, 48

Compact section
 See Classification of sections

Compressive strength
 AASHTO, 83
 ASD, 23
 ASD (allowable), 23
 BS, 119
 CISC, 100
 Eurocode, 139
 LRFD, 54

Design codes, 1
 See Also "Supported design codes"

Design load combinations, 6

Design output, 151
 graphical, 152
 member specific, 154
 tabular, 153

Design stations, 7

Effective length factor, 10

Euler buckling load
 AASHTO, 82
 ASD, 24

- BS, 119
- CISC, 100
- Eurocode, 139
- LRFD, 52
- Factored forces and moments
 - AASHTO, 79
 - BS, 117
 - CISC, 97
 - Eurocode, 137
 - LRFD, 52
- Flexural buckling
 - AASHTO, 83
 - ASD, 23
 - BS, 119
 - CISC, 100
 - Eurocode, 139
 - LRFD, 23, 54
- Graphical output, 152
- Interaction equations
 - See Capacity ratio
- Interactive environment, 1
- Lateral drift effect, 8
 - See Also P-Delta analysis
- Lateral-torsional buckling
 - AASHTO, 88
 - ASD, 30
 - BS, 122
 - CISC, 101
 - Eurocode, 143
 - LRFD, 61, 66, 69
- Live load reduction factor, 7, 18, 48, 79, 96, 114, 132
- Loading combinations, 2
 - AASHTO, 78
 - ASD, 18
 - BS, 114
 - CISC, 96
 - Eurocode, 132
- LRFD, 48
- Member specific output, 154
- Member stability effect, 8
 - See Also P-Delta analysis
- Moment magnification
 - AASHTO, 82
 - BS, 117
 - CISC, 97
 - Eurocode, 138
 - LRFD, 52
- Noncompact section
 - See Classification of sections
- Nonsway, 8
 - AASHTO, 79
 - BS, 119
 - CISC, 97
 - Eurocode, 137
 - LRFD, 52
- Notional load
 - BS, 114
 - CISC, 96
 - Eurocode, 132
- Output, 2
 - details, 155
 - graphical, 151
 - tabular, 151
- P-Delta analysis, 8
 - AASHTO, 79, 82
 - BS, 114, 119
 - CISC, 96 - 97
 - Eurocode, 133, 138
 - LRFD, 48, 53
- P-Delta effects, 8
- Perry factor, 119
- Plastic section
 - See Classification of sections

Redesign, 155

Robertson constant, 119

Second order effects
See P-Delta effects

Shear strength
AASHTO, 90
ASD (allowable), 39
BS, 125
CISC, 105
Eurocode, 141
LRFD, 72

Slender section
See Classification of sections

Strength reduction factors
AASHTO, 82
BS (partial factors), 119
CISC, 100
Euro (partial factors), 138
LRFD, 54

Supported design codes, 1
AASHTO, 5, 75
ASD, 5, 15
BS, 5, 111
CISC, 5, 93
Eurocode, 5, 129
LRFD, 5, 45

Sway, 8
AASHTO, 79
BS, 119
CISC, 97
Eurocode, 137
LRFD, 52

Tabular output, 153

Tensile strength
AASHTO, 84
ASD (allowable), 23
BS, 121
CISC, 101

Eurocode, 139
LRFD, 60

Unbraced frames, 8
AASHTO, 79
BS, 119
CISC, 97
Eurocode, 137
LRFD, 52

Units, 2, 13
AASHTO, 78
ASD, 18
BS, 111
CISC, 93
Eurocode, 129
LRFD, 48

Unsupported length, 9

DYNAMIC ANALYSIS

*Force Equilibrium Is Fundamental In
The Dynamic Analysis Of Structures*

12.1 INTRODUCTION

All real physical structures, when subjected to loads or displacements, behave dynamically. The additional inertia forces, *from Newton's second law*, are equal to the mass times the acceleration. If the loads or displacements are applied very slowly then the inertia forces can be neglected and a static load analysis can be justified. Hence, dynamic analysis is a simple extension of static analysis.

In addition, all real structures potentially have an infinite number of displacements. Therefore, the most critical phase of a structural analysis is to create a computer model, with a finite number of massless members and a finite number of node (joint) displacements, that will simulate the behavior of the real structure. The mass of a structural system, which can be accurately estimated, is lumped at the nodes. Also, for linear elastic structures the stiffness properties of the members, with the aid of experimental data, can be approximated with a high degree of confidence. However, the dynamic loading, energy dissipation properties and boundary (foundation) conditions for many structures are difficult to estimate. This is always true for the cases of seismic input or wind loads.

To reduce the errors that may be caused by the approximations summarized in the previous paragraph, it is necessary to conduct many different dynamic analyses using different computer models, loading and boundary conditions. It is not unrealistic to conduct 20 or more computer runs to design a new structure or to investigate retrofit options for an existing structure.

Because of the large number of computer runs required for a typical dynamic analysis, it is very important that accurate and numerically efficient methods be used within computer programs. Some of these methods have been developed by the author and are relatively new. Therefore, one of the purposes of this book is to summarize these numerical algorithms, their advantages and limitations.

12.2 DYNAMIC EQUILIBRIUM

The force equilibrium of a multi-degree-of-freedom lumped mass system as a function of time can be expressed by the following relationship:

$$\mathbf{F}(t)_I + \mathbf{F}(t)_D + \mathbf{F}(t)_S = \mathbf{F}(t) \quad (12.1)$$

in which the force vectors at time t are

- $\mathbf{F}(t)_I$ is a vector of inertia forces acting on the node masses
- $\mathbf{F}(t)_D$ is a vector of viscous damping, or energy dissipation, forces
- $\mathbf{F}(t)_S$ is a vector of internal forces carried by the structure
- $\mathbf{F}(t)$ is a vector of externally applied loads

Equation (12.1) is based on physical laws and is valid for both linear and nonlinear systems if equilibrium is formulated with respect to the deformed geometry of the structure.

For many structural systems, the approximation of linear structural behavior is made in order to convert the physical equilibrium statement, Equation (12.1), to the following set of second-order, linear, differential equations:

$$\mathbf{M} \ddot{\mathbf{u}}(t)_a + \mathbf{C} \dot{\mathbf{u}}(t)_a + \mathbf{K} \mathbf{u}(t)_a = \mathbf{F}(t) \quad (12.2)$$

in which \mathbf{M} is the mass matrix (lumped or consistent), \mathbf{C} is a viscous damping matrix (which is normally selected to approximate energy dissipation in the real structure) and \mathbf{K} is the static stiffness matrix for the system of structural elements. The time-dependent vectors $\mathbf{u}(t)_a$, $\dot{\mathbf{u}}(t)_a$ and $\ddot{\mathbf{u}}(t)_a$ are the absolute node displacements, velocities and accelerations, respectively.

Many books on structural dynamics present several different methods of applied mathematics to obtain the exact solution of Equation (12.2). Within the past several years, however, with the general availability of inexpensive, high-speed personal computers (see Appendix Z) the exact solution of Equation (12.2) can be obtained without the use of complex mathematical techniques. Therefore, the modern structural engineer, with a physical understanding of dynamic equilibrium and energy dissipation, can perform dynamic analysis of complex structural systems. A strong engineering mathematical background is desirable; however, in my opinion, it is no longer mandatory.

For seismic loading, the external loading $\mathbf{F}(t)$ is equal to zero. The basic seismic motions are the three components of free-field ground displacements $u(t)_{ig}$ that are known at some point below the foundation level of the structure. Therefore, we can write Equation (12.2) in terms of the displacements $\mathbf{u}(t)$, velocities $\dot{\mathbf{u}}(t)$ and accelerations $\ddot{\mathbf{u}}(t)$ that are relative to the three components of free-field ground displacements.

Therefore, the absolute displacements, velocities and accelerations can be eliminated from Equation (12.2) by writing the following simple equations:

$$\begin{aligned}\mathbf{u}(t)_a &= \mathbf{u}(t) + \mathbf{I}_x u(t)_{xg} + \mathbf{I}_y u(t)_{yg} + \mathbf{I}_z u(t)_{zg} \\ \dot{\mathbf{u}}(t)_a &= \dot{\mathbf{u}}(t) + \mathbf{I}_x \dot{u}(t)_{xg} + \mathbf{I}_y \dot{u}(t)_{yg} + \mathbf{I}_z \dot{u}(t)_{zg} \\ \ddot{\mathbf{u}}(t)_a &= \ddot{\mathbf{u}}(t) + \mathbf{I}_x \ddot{u}(t)_{xg} + \mathbf{I}_y \ddot{u}(t)_{yg} + \mathbf{I}_z \ddot{u}(t)_{zg}\end{aligned}\quad (12.3)$$

where \mathbf{I}_i is a vector with ones in the “ i ” directional degrees-of-freedom and zero in all other positions. The substitution of Equation (12.3) into Equation (12.2) allows the node point equilibrium equations to be rewritten as

$$\mathbf{M}\ddot{\mathbf{u}}(t) + \mathbf{C}\dot{\mathbf{u}}(t) + \mathbf{K}\mathbf{u}(t) = -\mathbf{M}_x\ddot{u}(t)_{xg} - \mathbf{M}_y\ddot{u}(t)_{yg} - \mathbf{M}_z\ddot{u}(t)_{zg} \quad (12.4)$$

where $\mathbf{M}_i = \mathbf{M}\mathbf{I}_i$.

The simplified form of Equation (12.4) is possible since the rigid body velocities and displacements associated with the base motions cause no additional damping or structural forces to be developed.

It is important for engineers to realize that the displacements, which are normally printed by a computer program, are relative displacements and that the fundamental loading on the structure is foundation displacements and not externally applied loads at the joints of the structure. For example, the static pushover analysis of a structure is a poor approximation of the dynamic behavior of a three dimensional structure subjected to complex time-dependent base motions. Also, one must calculate absolute displacements to properly evaluate base isolation systems.

There are several different classical methods that can be used for the solution of Equation (12.4). Each method has advantages and disadvantages that depend on the type of structure and loading. To provide a general background for the various topics presented in this book, the different numerical solution methods are summarized below.

12.3 STEP BY STEP SOLUTION METHOD

The most general solution method for dynamic analysis is an incremental method in which the equilibrium equations are solved at times Δt , $2\Delta t$, $3\Delta t$, etc. There are a large number of different incremental solution methods. In general, they involve a solution of the complete set of equilibrium equations at each time increment. In the case of nonlinear analysis, it may be necessary to reform the stiffness matrix for the complete structural system for each time step. Also, iteration may be required within each time increment to satisfy equilibrium. As a result of the large computational requirements it can take a significant amount of time to solve structural systems with just a few hundred degrees-of-freedom.

In addition, artificial or numerical damping must be added to most incremental solution methods in order to obtain stable solutions. For this reason, engineers must be very careful in the interpretation of the results. For some nonlinear structures, subjected to seismic motions, incremental solution methods are necessary.

For very large structural systems, a combination of mode superposition and incremental methods has been found to be efficient for systems with a small number

of nonlinear members. This method has been incorporated in the new versions of SAP and ETABS and will be presented in detail later in this book.

12.4 MODE SUPERPOSITION METHOD

The most common and effective approach for seismic analysis of linear structural systems is the mode superposition method. This method, after a set of orthogonal vectors are evaluated, reduces the large set of global equilibrium equations to a relatively small number of uncoupled second order differential equations. The numerical solution of these equations involves greatly reduced computational time.

It has been shown that seismic motions excite only the lower frequencies of the structure. Typically, earthquake ground accelerations are recorded at increments of 200 points per second. Therefore, the basic loading data does not contain information over 50 cycles per second. Hence, neglecting the higher frequencies and mode shapes of the system normally does not introduce errors.

12.5 RESPONSE SPECTRA ANALYSIS

The basic mode superposition method, which is restricted to linearly elastic analysis, produces the complete time history response of joint displacements and member forces due to a specific ground motion loading [1,2]. There are two major disadvantages of using this approach. First, the method produces a large amount of output information that can require an enormous amount of computational effort to conduct all possible design checks as a function of time. Second, the analysis must be repeated for several different earthquake motions in order to assure that all the significant modes are excited, since a response spectrum for one earthquake, in a specified direction, is not a smooth function.

There are significant computational advantages in using the response spectra method of seismic analysis for prediction of displacements and member forces in structural systems. The method involves the calculation of only the maximum values of the displacements and member forces in each mode using smooth design spectra that are the average of several earthquake motions. In this book, we will recommend the CQC method to combine these maximum modal response values to obtain the most probable peak value of displacement or force. In addition, it will be shown that the

SRSS and CQC3 methods of combining results from orthogonal earthquake motions will allow one dynamic analysis to produce design forces for all members in the structure.

12.6 SOLUTION IN THE FREQUENCY DOMAIN

The basic approach, used to solve the dynamic equilibrium equations in the frequency domain, is to expand the external loads $\mathbf{F}(t)$ in terms of Fourier series or Fourier integrals. The solution is in terms of complex numbers that cover the time span from $-\infty$ to ∞ . Therefore, it is very effective for periodic types of loads such as mechanical vibrations, acoustics, sea-waves and wind [1]. However, the use of the frequency domain solution method for solving structures subjected to earthquake motions has the following disadvantages:

1. The mathematics, for most structural engineers including myself, is difficult to understand. Also, the solutions are difficult to verify.
2. Earthquake loading is not periodic; therefore, it is necessary to select a long time period in order that the solution from a finite length earthquake is completely damped out prior to the application of the same earthquake at the start of the next period of loading.
3. For seismic type loading the method is not numerically efficient. The transformation of the result from the frequency domain to the time domain, even with the use of Fast Fourier Transformation methods, requires a significant amount of computational effort.
4. The method is restricted to the solution of linear structural systems.
5. The method has been used, without sufficient theoretical justification, for the approximate nonlinear solution of site response problems and soil/structure interaction problems. Typically, it is used in an iterative manner to create linear equations. The linear damping terms are changed after each iteration in order to approximate the energy dissipation in the soil. Hence, dynamic equilibrium, within the soil, is not satisfied.

12.7 SOLUTION OF LINEAR EQUATIONS

The step-by-step solution of the dynamic equilibrium equations, the solution in the frequency domain, and the evaluation of eigenvectors and Ritz vectors all require the solution of linear equations of the following form:

$$\mathbf{AX} = \mathbf{B} \quad (12.5)$$

Where \mathbf{A} is an N by N symmetric matrix which contains a large number of zero terms. The N by M \mathbf{X} displacement and \mathbf{B} load matrices indicate that more than one load condition can be solved at the same time.

The method used in many computer programs, including SAP2000 [5] and ETABS [6], is based on the profile or active column method of compact storage. Because the matrix is symmetric, it is only necessary to form and store the first nonzero term in each column down to the diagonal term in that column. Therefore, the sparse square matrix can be stored as a one dimensional array along with a N by 1 integer array that indicates the location of each diagonal term. If the stiffness matrix exceeds the high-speed memory capacity of the computer a block storage form of the algorithm exists. Therefore, the capacity of the solution method is governed by the low speed disk capacity of the computer. This solution method is presented in detail in Appendix C of this book.

12.8 UNDAMPED HARMONIC RESPONSE

The most common and very simple type of dynamic loading is the application of steady-state harmonic loads of the following form:

$$\mathbf{F}(t) = \mathbf{f} \sin(\bar{\omega} t) \quad (12.5)$$

The node point distribution of all static load patterns, \mathbf{f} , which are not a function of time, and the frequency of the applied loading, $\bar{\omega}$, are user specified. Therefore, for the case of zero damping, the exact node point equilibrium equations for the structural system are

$$\mathbf{M}\ddot{\mathbf{u}}(t) + \mathbf{K}\mathbf{u}(t) = \mathbf{f} \sin(\bar{\omega} t) \quad (12.6)$$

The exact steady-state solution of this equation requires that the node point displacements and accelerations are given by

$$u(t) = v \sin(\bar{\omega} t), \quad \ddot{u}(t) = -v \bar{\omega}^2 \sin(\bar{\omega} t) \quad (12.7)$$

Therefore, the harmonic node point response amplitude is given by the solution of the following set of linear equations:

$$[K - \bar{\omega}^2 M]v = f \quad \text{or} \quad \bar{K}v = f \quad (12.8)$$

It is of interest to note that the normal solution for static loads is nothing more than a solution of this equation for zero frequency for all loads. It is apparent that the computational effort required for the calculation of undamped steady-state response is almost identical to that required by a static load analysis. Note that it is not necessary to evaluate mode shapes or frequencies to solve for this very common type of loading. The resulting node point displacements and member forces vary as $\sin(\bar{\omega} t)$. However, other types of loads that do not vary with time, such as dead loads, must be evaluated in a separate computer run.

12.9 UNDAMPED FREE VIBRATIONS

Most structures are in a continuous state of dynamic motion because of random loading such as wind, vibrating equipment, or human loads. These small ambient vibrations are normally near the natural frequencies of the structure and are terminated by energy dissipation in the real structure. However, special instruments attached to the structure can easily measure the motion. Ambient vibration field tests are often used to calibrate computer models of structures and their foundations.

After all external loads are removed from the structure, the equilibrium equation, which governs the undamped free vibration of a typical displaced shape v , is

$$M\ddot{v} + Kv = 0 \quad (12.9)$$

At any time the displaced shape v may be a natural mode shape of the system, or any combination of the natural mode shapes. However, it is apparent the total energy within an undamped free vibrating system is a constant with respect to time. The sum of the kinetic energy and strain energy, at all points in time, is a constant

and is defined as the *mechanical energy* of the dynamic system and can be calculated from:

$$E_M = \frac{1}{2} \dot{\mathbf{v}}^T \mathbf{M} \dot{\mathbf{v}} + \frac{1}{2} \mathbf{v}^T \mathbf{K} \mathbf{v} \quad (12.10)$$

12.10 SUMMARY

Dynamic analysis of three dimensional structural systems is a direct extension of static analysis. The elastic stiffness matrices are the same for both dynamic and static analysis. It is only necessary to lump the mass of the structure at the joints. The addition of inertia forces and energy dissipation forces will satisfy dynamic equilibrium. The dynamic solution for steady state harmonic loading, without damping, involves the same numerical effort as a static solution. Classically, there are many different mathematical methods to solve the dynamic equilibrium equations. However, it will later be shown in this book that the majority of both linear and nonlinear systems can be solved with one numerical method.

Energy is fundamental in dynamic analysis. At any point in time the external work supplied to the system must be equal to the sum of the kinetic and strain energy plus the energy dissipated in the system.

It is my opinion, with respect to earthquake resistant design, that we should try to minimize the mechanical energy in the structure. It is apparent that a rigid structure will have only kinetic energy and zero strain energy. On the other hand, a completely base isolated structure will have zero kinetic energy and zero strain energy. A structure cannot fail if it has zero strain energy.

12.11 REFERENCES

1. R. Clough, and J. Penzien, *Dynamics of Structures*, Second Edition, McGraw-Hill, Inc., ISBN 0-07-011394-7, 1993.
2. A. Chopra, *Dynamics of Structures*, Prentice-Hall, Inc., Englewood Cliffs, New Jersey 07632, ISBN 0-13-855214-2, 1995.
3. K. Bathe, *Finite Element Procedures in Engineering Analysis*, Prentice-Hall, Inc., Englewood Cliffs, New Jersey 07632, ISBN 0-13-317305-4, 1982.
4. E. L. Wilson and K. Bathe, "Stability and Accuracy Analysis of Direct Integration Methods," *Earthquake Engineering and Structural Dynamics*, Vol. 1, pp. 283-291, 1973.
5. *SAP2000 - Integrated Structural Analysis & Design Software*, Computers and Structures, Inc., Berkeley, California, 1997.
6. A. Habibullah, *ETABS - Three Dimensional Analysis of Building Systems, Users Manual*, Computers and Structures Inc., Berkeley, California, 1997.

SEISMIC ANALYSIS MODELING TO SATISFY BUILDING CODES

*The Current Building Codes Use the Terminology
Principal Direction without A Unique Definition*

17.1. INTRODUCTION

Currently a three-dimensional dynamic analysis is required for a large number of different types of structural systems that are constructed in Seismic Zones 2, 3 and 4 [1]. The lateral force requirements suggest several methods that can be used to determine the distribution of seismic forces within a structure. However, these guidelines are not unique and need further interpretations.

The major advantage of using the forces obtained from a dynamic analysis as the basis for a structural design is that the vertical distribution of forces may be significantly different from the forces obtained from an equivalent static load analysis. Consequently, the use of dynamic analysis will produce structural designs that are more earthquake resistant than structures designed using static loads.

For many years, approximate two-dimensional static load was acceptable as the basis for seismic design in many geographical areas and for most types of structural systems. During the past twenty years, due to the increasing availability of modern digital computers, most engineers have had experience with the static load analysis of three dimensional structures. However, few engineers, and the writers of the current building code, have had experience with the three dimensional dynamic

response analysis. Therefore, the interpretation of the dynamic analysis requirement of the current code represents a new challenge to most structural engineers.

The current code allows the results obtained from a dynamic analysis to be normalized so that the maximum dynamic base shear is equal to the base shear obtained from a simple two-dimensional static load analysis. Most members of the profession realize that there is no theoretical foundation for this approach. However, for the purpose of selecting the magnitude of the dynamic loading that will satisfy the code requirements, this approach can be accepted, in a modified form, until a more rational method is adopted.

The calculation of the "design base shears" is simple and the variables are defined in the code. It is of interest to note, however, that the basic magnitude of the seismic loads has not changed significantly from previous codes. The major change is that "dynamic methods of analysis" must be used in the "principal directions" of the structure. The present code does not state how to define the principal directions for a three dimensional structure of arbitrary geometric shape. Since the design base shear can be different in each direction, this "scaled spectra" approach can produce a different input motion for each direction, for both regular and irregular structures. Therefore, *the current code dynamic analysis approach can result in a structural design which is relatively "weak" in one direction.* The method of dynamic analysis proposed in this chapter results in a structural design that has equal resistance in all directions.

In addition, the maximum possible design base shear, which is defined by the present code, is approximately 35 percent of the weight of the structure. For many structures, it is less than 10 percent. It is generally recognized that this force level is small when compared to measured earthquake forces. Therefore, the use of this design base shear requires that substantial ductility be designed into the structure.

The definition of an irregular structure, the scaling of the dynamic base shears to the static base shears for each direction, the application of accidental torsional loads and the treatment of orthogonal loading effects are areas which are not clearly defined in the current building code. The purpose of this section is to present one method of three dimensional seismic analysis that will satisfy the Lateral Force Requirements of the code. The method is based on the response spectral shapes defined in the code and previously published and accepted computational procedures.

17.2. THREE DIMENSIONAL COMPUTER MODEL

Real and accidental torsional effects must be considered for all structures. Therefore, all structures must be treated as three dimensional systems. Structures with irregular plans, vertical setbacks or soft stories will cause no additional problems if a realistic three dimensional computer model is created. This model should be developed in the very early stages of design since it can be used for static wind and vertical loads, as well as dynamic seismic loads.

Only structural elements with significant stiffness and ductility should be modeled. Non-structural brittle components can be neglected. However, shearing, axial deformations and non-center line dimensions can be considered in all members without a significant increase in computational effort by most modern computer programs. The rigid, in-plane approximation of floor systems has been shown to be acceptable for most buildings. For the purpose of elastic dynamic analysis, gross concrete sections, neglecting the stiffness of the steel, are normally used. A cracked section mode should be used to check the final design.

The P-Delta effects should be included in all structural models. It has been shown in Chapter 11 that these second order effects can be considered, without iteration, for both static and dynamic loads. The effect of including P-Delta displacements in a dynamic analysis results in a small increase in the period of all modes. In addition to being more accurate, an additional advantage of automatically including P-Delta effects is that the moment magnification factor for all members can be taken as unity in all subsequent stress checks.

The mass of the structure can be estimated with a high degree of accuracy. The major assumption required is to estimate the amount of live load to be included as added mass. For certain types of structures it may be necessary to conduct several analyses with different values of mass. The lumped mass approximation has proven to be accurate. In the case of the rigid diaphragm approximation, the rotational mass moment of inertia must be calculated.

The stiffness of the foundation region of most structures can be modeled by massless structural elements. It is particularly important to model the stiffness of piles and the rotational stiffness at the base of shear walls.

The computer model for static loads only should be executed prior to conducting a dynamic analysis. Equilibrium can be checked and various modeling approximations can be verified with simple static load patterns. The results of a dynamic analysis are generally very complex and the forces obtained from a response spectra analysis are always positive. Therefore, dynamic equilibrium is almost impossible to check. However, it is relatively simple to check energy balances in both linear and nonlinear analysis.

17.3. THREE DIMENSIONAL MODE SHAPES AND FREQUENCIES

The first step in the dynamic analysis of a structural model is the calculation of the three dimensional mode shapes and natural frequencies of vibration. Within the past several years, very efficient computational methods have been developed which have greatly decreased the computational requirements associated with the calculation of orthogonal shape functions as presented in Chapter 14. It has been demonstrated that load-dependent Ritz vectors, which can be generated with a minimum of numerical effort, produce more accurate results when used for a seismic dynamic analysis than if the exact free-vibration mode shapes are used.

Therefore, a dynamic response spectra analysis can be conducted with approximately twice the computer time requirements of a static load analysis. Since systems with over 60,000 dynamic degrees-of-freedom can be solved within a few hours on personal computers, there is not a significant increase in cost between a static and a dynamic analysis. The major cost is the "man hours" required to produce the three dimensional computer model that is necessary for a static or a dynamic analysis.

In order to illustrate the dynamic properties of the three dimensional structure, the mode shapes and frequencies are calculated for the irregular, eight story, 80 foot tall building shown in Figure 17.1. This building is a concrete structure with several hundred degrees-of-freedom. However, the three components of mass are lumped at each of the eight floor levels. Therefore, only 24 three dimensional mode shapes are possible.

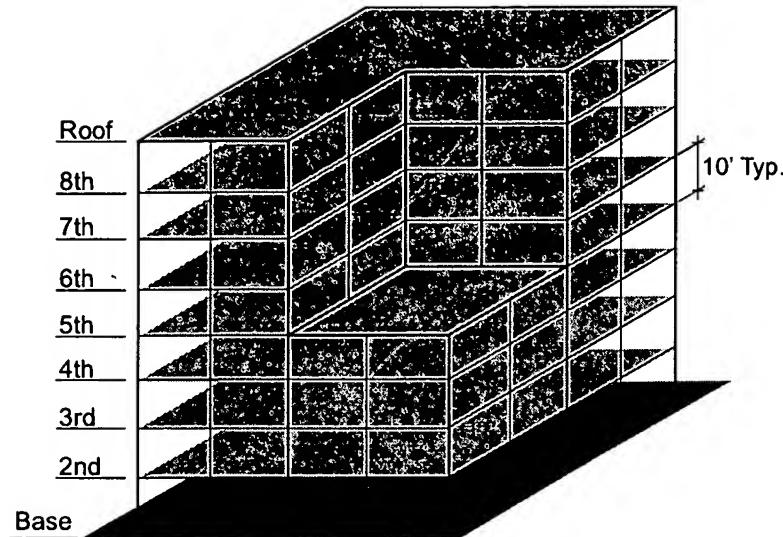


Figure 17.1. Example of Eight Story Irregular Building

Each three dimensional mode shape of a structure may have displacement components in all directions. For the special case of a symmetrical structure, the mode shapes are uncoupled and will have displacement in one direction only. Since each mode can be considered to be a deflection due to a set of static loads, six base reaction forces can be calculated for each mode shape. For the structure shown in Figure 17.1, Table 17.1 summarizes the two base reactions and three overturning moments associated with each mode shape. Since vertical mass has been neglected there is no vertical reaction. The magnitudes of the forces and moments have no meaning since the amplitude of a mode shape can be normalized to any value. However, the relative values of the different components of the shears and moments associated with each mode are of considerable value. The modes with a large torsional component are highlighted in **bold**.

Table 17.1. Three Dimensional Base Forces and Moments

MODE	PERIOD	MODAL BASE SHEAR REACTIONS			MODAL OVERTURNING MOMENTS		
		Seconds	X-DIR	Y-DIR	Angle Deg.	X-AXIS	Y-AXIS
1	.6315	.781	.624	38.64	-37.3	46.6	-18.9
2	.6034	-.624	.781	-51.37	-46.3	-37.0	38.3
3	.3501	.785	.620	38.30	-31.9	40.2	85.6
4	.1144	-.753	-.658	41.12	12.0	-13.7	7.2
5	.1135	.657	-.754	-48.89	13.6	11.9	-38.7
6	.0706	.989	.147	8.43	-33.5	51.9	2438.3
7	.0394	-.191	.982	-79.01	-10.4	-2.0	29.4
8	.0394	-.983	-.185	10.67	1.9	-10.4	26.9
9	.0242	.848	.530	32.01	-5.6	8.5	277.9
10	.0210	.739	.673	42.32	-5.3	5.8	-3.8
11	.0209	.672	-.740	-47.76	5.8	5.2	-39.0
12	.0130	-.579	.815	-54.63	-.8	-8.8	-1391.9
13	.0122	.683	.730	46.89	-4.4	4.1	-6.1
14	.0122	.730	-.683	-43.10	4.1	4.4	-40.2
15	.0087	-.132	-.991	82.40	5.2	-.7	-22.8
16	.0087	-.991	.135	-7.76	-.7	-5.2	30.8
17	.0074	-.724	-.690	43.64	4.0	-4.2	-252.4
18	.0063	-.745	-.667	41.86	3.1	-3.5	7.8
19	.0062	-.667	.745	-48.14	-3.5	-3.1	38.5
20	.0056	-.776	-.630	39.09	2.8	-3.4	54.1
21	.0055	-.630	.777	-50.96	-3.4	-2.8	38.6
22	.0052	.776	.631	39.15	-2.9	3.5	66.9
23	.0038	-.766	-.643	40.02	3.0	-3.6	-323.4
24	.0034	-.771	-.637	39.58	2.9	-3.5	-436.7

A careful examination of the directional properties of the three dimensional mode shapes at the early stages of a preliminary design can give a structural engineer additional information which can be used to improve the earthquake resistant design of a structure. The current code defines an "irregular structure" as one which has a certain geometric shape or in which stiffness and mass discontinuities exist. A far

more rational definition is that a "regular structure" is one in which there is a minimum coupling between the lateral displacements and the torsional rotations for the mode shapes associated with the lower frequencies of the system. Therefore, if the model is modified and "tuned" by studying the three dimensional mode shapes during the preliminary design phase, it may be possible to convert a "geometrically irregular" structure to a "dynamically regular" structure from an earthquake-resistant design standpoint.

Table 17.2. Three Dimensional Participating Mass - (percent)

MODE	X-DIR	Y-DIR	Z-DIR	X-SUM	Y-SUM	Z-SUM
1	34.224	21.875	.000	34.224	21.875	.000
2	23.126	36.212	.000	57.350	58.087	.000
3	2.003	1.249	.000	59.354	59.336	.000
4	13.106	9.987	.000	72.460	69.323	.000
5	9.974	13.102	.000	82.434	82.425	.000
6	.002	.000	.000	82.436	82.425	.000
7	.293	17.770	.000	82.729	90.194	.000
8	7.726	.274	.000	90.455	90.469	.000
9	.039	.015	.000	90.494	90.484	.000
10	2.382	1.974	.000	92.876	92.458	.000
11	1.955	2.370	.000	94.831	94.828	.000
12	.000	.001	.000	94.831	94.829	.000
13	1.113	1.271	.000	95.945	96.100	.000
14	1.276	1.117	.000	97.220	97.217	.000
15	.028	1.556	.000	97.248	98.773	.000
16	1.555	.029	.000	98.803	98.802	.000
17	.011	.010	.000	98.814	98.812	.000
18	.503	.403	.000	99.316	99.215	.000
19	.405	.505	.000	99.722	99.720	.000
20	.102	.067	.000	99.824	99.787	.000
21	.111	.169	.000	99.935	99.957	.000
22	.062	.041	.000	99.997	99.998	.000
23	.003	.002	.000	100.000	100.000	.000
24	.001	.000	.000	100.000	100.000	.000

For this building, it is of interest to note that the mode shapes, which tend to have directions that are 90 degrees apart, have almost the same value for their period. This is typical of three dimensional mode shapes for both regular and irregular buildings. For regular symmetric structures, which have equal stiffness in all directions, the periods associated with the lateral displacements will result in pairs of identical periods. However, the directions associated with the pair of three dimensional mode shapes are not mathematically unique. For identical periods, most computer programs allow round-off errors to produce two mode shapes with directions which differ by 90 degrees. Therefore, the SRSS method should not be used to combine modal maximums in three dimensional dynamic analysis. The CQC method eliminates problems associated with closely spaced periods.

For a response spectrum analysis, the current code states that "at least 90 percent of the participating mass of the structure must be included in the calculation of response for each principal direction." Therefore, the number of modes to be evaluated must satisfy this requirement. Most computer programs automatically calculate the participating mass in all directions using the equations presented in Chapter 13. This requirement can be easily satisfied using LDR vectors. For the structure shown in Figure 17.1, the participating mass for each mode and for each direction is shown in Table 17.2. For this building, only eight modes are required to satisfy the 90 percent specification in both the x and y directions.

17.4. THREE DIMENSIONAL DYNAMIC ANALYSIS

It is possible to conduct a dynamic, time-history, response analysis by either the mode superposition or step-by-step methods of analysis. However, a standard time-history ground motion, for the purpose of design, has not been defined. Therefore, most engineers use the response spectrum method of analysis as the basic approach. The first step in a response spectrum analysis is the calculation of the three dimensional mode shapes and frequencies as indicated in the previous section.

17.4.1. Dynamic Design Base Shear

For dynamic analysis, the 1994 UBC requires that the "design base shear", V , is to be evaluated from the following formula:

$$V = [Z I C / R_w] \bar{W} \quad (17.1)$$

Where

Z = Seismic zone factor given in Table 16-I.

I = Importance factor given in Table 16-K.

R_w = Numerical coefficient given in Table 16-N or 16-P.

W = The total seismic weight of the structure.

C = Numerical coefficient (2.75 maximum value) determined from:

$$C = 1.25 S / T^{2/3} \quad (1-2)$$

Where

S = Site coefficient for soil characteristics given in Table 16-J.

T = Fundamental period of vibration (seconds).

The period, **T**, determined from the three dimensional computer model, can be used for most cases. This is essentially Method B of the code.

Since the computer model often neglects nonstructural stiffness, the code requires that Method A be used under certain conditions. Method A defines the period, **T**, as follows:

$$T = C_t h^{3/4} \quad (1-3)$$

where **h** is the height of the structure in feet and **C_t** is defined by the code for various types of structural systems.

The Period calculated by Method B cannot be taken as more than 30% longer than that computed using Method A in Seismic Zone 4 and more than 40% longer in Seismic Zones 1, 2 and 3.

For a structure that is defined by the code as "regular", the design base shear may be reduced by an additional 10 percent. However, it must not be less than 80 percent of the shear calculated using Method A. For an "irregular" structure this reduction is not allowed.

17.4.2. Definition of Principal Directions

A weakness in the current code is the lack of definition of the “principal horizontal directions” for a general three dimensional structure. If each engineer is allowed to select an arbitrary reference system, the “dynamic base shear” will not be unique and each reference system could result in a different design. One solution to this problem, that will result in a unique design base shear, is to use the direction of the base shear associated with the fundamental mode of vibration as the definition of the “major principal direction” for the structure. The “minor principal direction” will be, by definition, ninety degrees from the major axis. This approach has some rational basis since it is valid for regular structures. Therefore, this definition of the principal directions will be used for the method of analysis presented in this chapter.

17.4.3. Directional and Orthogonal Effects

The required design seismic forces may come from any horizontal direction and, for the purpose of design, they may be assumed to act non-concurrently in the direction of each principal axis of the structure. In addition, for the purpose of member design, the effects of seismic loading in two orthogonal directions may be combined on a square-root-of-the-sum-of-the-squares (SRSS) basis. (Also, it is allowable to design members for 100 percent of the seismic forces in one direction plus 30 percent of the forces produced by the loading in the other direction. We will not use this approach in the procedure suggested here for reasons presented in Chapter 15.)

17.4.4. Basic Method of Seismic Analysis

In order to satisfy the current requirements, it is necessary to conduct two separate spectrum analyses in the major and minor principal directions (as defined above). Within each of these analyses, the Complete Quadratic Combination (CQC) method is used to accurately account for modal interaction effects in the estimation of the maximum response values. The spectra used in both of these analyses can be obtained directly from the Normalized Response Spectra Shapes given by the Uniform Building Code.

17.4.5. Scaling of Results

Each of these analyses will produce a base shear in the major principal direction. A single value for the “dynamic base shear” is calculated by the SRSS method. Also,

a "dynamic base shear" can be calculated in the minor principal direction. The next step is to scale the previously used spectra shapes by the ratio of "design base shear" to the minimum value of the "dynamic base shear". This approach is more conservative than proposed by the current requirements, since only the scaling factor that produces the largest response is used. However, this approach is far more rational since it results in the same design earthquake in all directions.

17.4.6. Dynamic Displacements and Member Forces

The displacement and force distribution are calculated using the basic SRSS method to combine the results from 100 percent of the scaled spectra applied in each direction. If two analyses are conducted in any two orthogonal directions, in which the CQC method is used to combine the modal maximums for each analysis, and the results are combined by the SRSS method, exactly the same results will be obtained regardless of the orientation of the orthogonal reference system. Therefore, the direction of the base shear of the first mode defines a reference system for the building.

If site-specific spectra are given, for which scaling is not required, any orthogonal reference system can be used. In either case, only one computer run is necessary to calculate all member forces to be used for design.

17.4.7. Torsional Effects

Possible torsional ground motion, the unpredictable distribution of live load mass and the variations of structural properties are three reasons why both regular and irregular structures must be designed for accidental torsional loads. Also, for a regular structure lateral loads do not excite torsional modes. One method suggested in the Code is to conduct several different dynamic analyses with the mass at different locations. This approach is not practical since the basic dynamic properties of the structure (and the dynamic base shears) would be different for each analysis. In addition, the selection of the maximum member design forces would be a monumental post-processing problem.

The current Code allows the use of pure static torsional loads to predict the additional design forces caused by accidental torsion. The basic vertical distribution of lateral static loads is given by the Code equations. The static torsional moment at

any level is calculated by the multiplication of the static load at that level by 5 percent of the maximum dimension at that level. In this book it is recommended that these pure torsional static loads, applied at the center of mass at each level, be used as the basic approach to account for accidental torsional loads. This static torsional load is treated as a separate load condition so that it can be appropriately combined with the other static and dynamic loads.

17.5. NUMERICAL EXAMPLE

To illustrate the base-shear scaling method recommended here, a static seismic analysis is conducted on the building shown in Figure 17.1. The eight-story building has 10 feet story heights. The seismic dead load is 238.3 kips for the top four stories and 363.9 kips for the lower four stories. For $I = 1$, $Z = 0.4$, $S = 1.0$, and $R_w = 6.0$, the evaluation of Equation 17.1 yields the design base forces given in Table 17.3. Table 17.3. Static Design Base Forces Using The Uniform Building Code

Period (sec)	Angle (deg)	Base Shear	Overshooting Moment
0.631	38.64	279.9	14,533
0.603	-51.36	281.2	14,979

The normalized response spectra shape for soil type 1, which is defined in the Uniform Building Code, is used as the basic loading for the three dimensional dynamic analyses. Using eight modes only and the SRSS method of combining modal maxima, the base shears and overturning moments are summarized in Table 17.4 for various directions of loading.

Table 17.4. Dynamic Base Forces Using The SRSS Method

Angle -deg	BASE SHEARS		OVERTURNING MOMENTS	
	V₁	V₂	M₁	M₂
0	58.0	55.9	2982	3073
90	59.8	55.9	2983	3185
38.64	70.1	5.4	66	4135
-51.36	83.9	5.4	66	4500

The 1-axis is in the direction of the seismic input and the 2-axis is normal to the direction of the loading. This example clearly illustrates the major weakness of the SRSS method of modal combination. Unless the input is in the direction of the fundamental mode shapes, a large base shear is developed normal to the direction of the input and the dynamic base shear in the direction of the input is significantly underestimated as illustrated in Chapter 15.

As indicated by Table 17.5, the CQC method of modal combination eliminates problems associated with the SRSS method. Also, it clearly illustrates that the directions of 38.64 and -51.36 degrees are a good definition of the principal directions for this structure. Note that the directions of the base shears of the first two modes differ by 90.00 degrees.

Table 17.5. Dynamic Base Forces Using The CQC Method

Angle -deg	BASE SHEARS		OVERTURNING MOMENTS	
	V₁	V₂	M₁	M₂
0	78.1	20.4	1202	4116
90	79.4	20.4	1202	4199
38.64	78.5	0.2	3.4	4145
-51.36	84.2	0.2	3.4	4503

Table 17.6 summarizes the scaled dynamic base forces to be used as the basis for design by two different methods.

Table 17.6 Normalized Base Forces In Principal Directions

	38.64 Degrees		-51.36 Degrees	
	V (kips)	M(ft-kips)	V (kips)	M(ft-kips)
Static Code Forces	279.9	14,533	281.2	14,979
Dynamic Design Forces Scaled by Base Shear 279.9/78.5 = 3.57	279.9	14,732	299.2	16,004

For this case, the input spectra scale factor of 3.57 should be used for all directions and is based on the fact that both the dynamic base shears and the dynamic overturning moments must not be less than the static code forces. This approach is clearly more conservative than the approach suggested by the current Uniform Building Code. It is apparent that the use of different scale factors for a design spectra in the two different directions, as allowed by the code, results in a design that has a weak direction relative to the other principle direction.

17.6. DYNAMIC ANALYSIS METHOD SUMMARY

In this section, a dynamic analysis method is summarized that produces unique design displacements and member forces which will satisfy the current Uniform Building Code. It can be used for both regular and irregular structures. The major steps in the approach are as follows:

1. A three dimensional computer model must be created in which all significant structural elements are modeled. This model should be used in the early phases of design since it can be used for both static and dynamic loads.
2. The three dimensional mode shapes should be repeatedly evaluated during the design of the structure. The directional and torsional properties of the mode shapes can be used to improve the design. A well-designed structure should have a minimum amount of torsion in the mode shapes associated with the lower frequencies of the structure.

3. The direction of the base reaction of the mode shape associated with the fundamental frequency of the system is used to define the principal directions of the three dimensional structure.
4. The "design base shear" is based on the longest period obtained from the computer model, except when limited to 1.3 or 1.4 times the Method A calculated period.
5. Using the CQC method, the "dynamic base shears" are calculated in each principal direction due to 100 percent of the Normalized Spectra Shapes. Use the minimum value of the base shear in the principal directions to produce one "scaled design spectra".
6. The dynamic displacements and member forces are calculated using the SRSS value of 100 percent of the scaled design spectra applied non-concurrently in any two orthogonal directions as presented in Chapter 15.
7. A pure torsion static load condition is produced using the suggested vertical lateral load distribution defined in the code.
8. The member design forces are calculated using the following load combination rule:

$$F_{DESIGN} = F_{DEAD LOAD} \pm [F_{DYNAMIC} + |F_{TORSION}|] + F_{OTHER}$$

The dynamic forces are always positive and the accidental torsional forces must always increase the value of force. If vertical dynamic loads are to be considered, a dead load factor can be applied.

One can justify many other methods of analyses that will satisfy the current code. The approach presented in this chapter can be used directly with the computer programs ETABS and SAP2000 with their steel and concrete post-processors. Since these programs have very large capacities and operate on personal computers, it is possible for a structural engineer to investigate a large number of different designs very rapidly with a minimum expenditure of manpower and computer time.

17.7. SUMMARY

After being associated with the three dimensional dynamic analysis and design of a large number of structures during the past 40 years, the author would like to take this opportunity to offer some constructive comments on the lateral load requirements of the current code.

First: *the use of the "dynamic base shear" as a significant indication of the response of a structure may not be conservative.* An examination of the modal base shears and overturning moments in Tables 17.1 and 17.2 clearly indicates that base shears associated with the shorter periods produce relatively small overturning moments. Therefore, a dynamic analysis, which will contain higher mode response, will always produce a larger dynamic base shear relative to the dynamic overturning moment. Since the code allows all results to be scaled by the ratio of dynamic base shear to the static design base shear, the dynamic overturning moments can be significantly less than the results of a simple static code analysis. A scale factor based on the ratio of the "static design overturning moment" to the "dynamic overturning moment" would be far more logical. The static overturning moment can be calculated by using the static vertical distribution of the design base shear which is currently suggested in the code.

Second: *for irregular structures, the use of the terminology "period (or mode shape) in the direction under consideration" must be discontinued.* The stiffness and mass properties of the structure define the directions of all three dimensional mode shapes. The term "principal direction" should not be used unless it is clearly and uniquely defined.

Third: *the scaling of the results of a dynamic analysis should be re-examined.* The use of site-dependent spectra is encouraged.

Finally: *it is not necessary to distinguish between regular and irregular structures when a three dimensional dynamic analysis is conducted.* If an accurate three dimensional computer model is created, the vertical and horizontal irregularities and known eccentricities of stiffness and mass will cause the displacement and rotational components of the mode shapes to be coupled. A three dimensional dynamic analysis, based on these coupled mode shapes, will produce a far more complex response with larger forces than the response of a regular structure. It is possible to

predict the dynamic force distribution in a very irregular structure with the same degree of accuracy and reliability as the evaluation of the force distribution in a very regular structure. Consequently, if the design of an irregular structure is based on a realistic dynamic force distribution, there is no logical reason to expect that it will be any less earthquake resistant than a regular structure which was designed using the same dynamic loading. A reason why many irregular structures have a documented record of poor performance during earthquakes is that their designs were often based on approximate two dimensional static analyses.

One major advantage of the modeling method presented in this chapter is that one set of dynamic design forces, including the effects of accidental torsion, is produced with one computer run. Of greater significance, however, is the resulting structural design has equal resistance to seismic motions from all possible directions.

17.8. REFERENCES

1. *Recommended Lateral Force Requirements and Commentary, 1996 Sixth Edition*, Seismology Committee, Structural Engineers Association of California, Tel. 916-427-3647.

STRUCTURAL ANALYSIS WITH SAP2000**By Mr. Linzhong Deng and Prof. Michel Ghosn****Department of Civil Engineering****The City College of New York****CUNY**

SAP2000 is a general purpose finite element program which performs the static or dynamic, linear or nonlinear analysis of structural systems. It is also a powerful design tool to design structures following AASHTO specifications, ACI and AISC building codes. These features, and many more make SAP2000 the state-of-the-art in structural analysis program.

The SAP2000 graphic user interface (GUI) is used to model, analyze, design, and display the structure geometry, properties and analysis results. The analysis procedure can be divided into three parts:

1.
 1. Preprocessing.
 2. Solving.
 3. Postprocessing

Part I. Preprocessing.

In preprocessing, the following information is needed by SAP2000.

1.
 1. Choosing the units for this project.
 2. Setting up geometry.
 3. Defining material and member section properties.
 4. Assigning member section properties and element releases.
 5. Defining load cases.
 6. Assigning load magnitudes.
 7. Assigning restraints.

- I. Choosing units.

- - o From the combo (i.e. the drop down list) in the main window's status bar, choose the units

for this project.

II. Setting up structure's geometry.

There are two ways to set up the structure's geometry: The first is from the SAP2000's templates. The other is by creating a completely new model.

When creating from a template, follow these steps:

1.
 1. From File menu, choose New Model from Template... This will display the Model Template dialog box.
 2. In this dialog box:
 - a.
 - o
 -
 - b. Click on the template which most closely resembles the structure you want to analyze. This will display the template dialog box.
 - c. In this dialog box, choose the appropriate parameters.
 - c. Click OK button.

The screen will refresh and display 3-D and 2-D views of the model in vertically tiled adjoining windows. You can activate the one you plan to work in by clicking the window's title. You can any one of the two windows if you wish.

When creating from a new model, follow these steps:

1.
 1. From the File menu, choose the New Model... This will display the window of coordinate system definition.
 2. In the window of coordinate system definition, enter the appropriate grid information. The cross points of the grid will define the necessary joints of your structure. This will display the 3-D and 2-D view window with grid.
 3. 3-D and 2-D views of the model are displayed in vertically tiled adjoining windows. You can close the 3-D windows if you wish. Active the 2-D view by click the x-y button in toolbox or by clicking any point inside the 2-D window.
 4. From draw menu, choose "Draw From Element". This will change your mouse point from " " to " " in the area of 2-D view.
 5. Draw your structure in the grid based on the grid spacings defined in step 2. Click your left mouse button to define the joints. Every joint needs one click. SAP2000 will connect the joints automatically. Double click the left mouse button to stop the action of connection. When you draw something wrong, click the " " inside the floating toolbox situated in the lower part of your screen. Then click the members which need to be deleted. Then from the edit menu, choose "delete". To see the modified structure, from the display menu, choose the **show undeformed shape**.

III. Define material and structural section properties.

In this step, we are going to define all the material types and all section properties which are present in this structure. This requires the following steps:

1.

1. From the define menu, choose material... This will display the window of define material.
2. If your material is standard steel or concrete, you can click modify/show material button and use the library supplied properties. Click button **OK** to accept appropriate properties. Otherwise click the button add new material to define a new material's properties, or the button modify/show material to change the library's data according to your material's properties.
3. From the define menu, choose Frame sections... This will display the Frame sections dialog box.
4. In this dialog box, you can define a new section type, import a section's geometry from the SAP2000's library, or modify a section's geometry from the default values. Suppose you have two rectangular sections, you need the following steps to define these sections.

a.

o

- a. Highlight the **FSEC1** in the box frame name.
- b. Click the **modify/show section** button. This will display the window of rectangular section dialog box.
- c. In this box, choose the corresponding material from the material combo, type in the number in the section's width and height's text box. Click **ok** to terminate this dialog box and return to Frame sections dialog box.
- d. Choose **Add rectangular** in the second combo box. This will display the rectangular section dialog box.
- e. repeat step c to define the properties of section section.
- f. If you want to delete a section type, you highlight the section's name which is to be deleted, then click the button **delete sections**.

1.

1. Click on the **OK** button to return the main window.

IV. Assigning member section properties.

There are three selection methods used by SAP2000 to assign member properties, support restraints, loads... For clarity and convenience, these three selection methods are summoned here. The first is to click the members one by one after you click on the pointer tool button (" ") on the floating toolbox. The second way is to drag a rectangular box after you click on the pointer tool button on the floating toolbox. All of the objects inside this rectangular will be selected simultaneously. The third way is to draw a straight line after you click the "Sect intersecting line select mode" button on the floating toolbox. All the objects intersecting the line you draw will be simultaneously selected.

You need the following steps to assign member section properties:

a.

- a. Select a group of members which have the same sections by one of the 3 selection methods described above.
- b. From **assign** menu, choose **frame**, then **sections...** from the submenu. This will display the define frame sections dialog box.
- c. In the name area of this dialog box, click the section corresponding to this selected group (e.g. FSECT1 or FSECT2, etc).
- d. Repeat steps a, b and c until you have assigned a section for every member of the structure.
- e. Select a group of members which will be assigned the same member releases.
- f. From **assign** menu, choose **frame**, then **release...** from the submenu. This will display the frame release dialog box.
- g. Choose the appropriate release parameters for the already selected members. If these members are truss members, click the check-boxes of torsion-start, moment22-start, moment22-end, moment33-start and moment33-end.
- h. Repeat steps e, f and g until you finish to assign release properties for all the necessary members.

V. Defining load cases.

Now, it is time to give SAP2000 the applied load's information. The steps are:

a.

- a. From **Define** menu, choose **Static load cases...** This will display the define load case dialog box.
- b. This dialog box will display the default load, LOAD1, with type set to Dead, and self-weight multiplier set to unity. This will automatically include the self-weight of structural members in the analysis based on preset specific weights given in function of the material type. We don't have to change anything for this first load case. But if you wish to enter the weight by your self and put it as joint load, or if you want to ignore the offset of the dead weight, then you should change the self-weight multiplier to 0 to avoid count the self weight twice.
- c. Define additional load cases, change the LOAD1 to LOAD2 (or the case you defined), select load type from the Type drop-down list box, change the self-weight multiplier to appropriate number. In most times, you change the self-weight multiplier to 0 because dead load already count dead load in LOAD1). Then click on the **Add new Load** button to notify SAP2000. Repeat this step until you define all the load cases.
- d. Finally, click **OK** to back to main window.

In the following section of assigning joint load cases, you must assign a numerical volume and the location of each joint loads for every load cases.

VI. Assigning loads.

For simplicity, we just talk about assigning joint loads. If you wish to apply a distributed load on a member, you can refer to SAP2000 manual for detail. To assign joint loads execute the following steps:

a.

- a. Select the joints which have the same joint loads. You can use one of the three selection methods used previously to select members.
- b. From the **Assign** menu, choose **Joint Static Loads**, then **Forces...** from the submenu. This will display the Joint forces dialog box.
- c. In this dialog box, accept the default load case name as LOAD1, enter the corresponding joint force components in the Load area. Click **OK** to accept the above joint loads.
- d. Repeat steps a, b and c until you assign all the joint loads of this load case defined to this structure.
- e. Repeat steps a, b, c and d until you finish every load case's load assignment.

VII. Assigning restraints.

It is very important to assign restraints to your structure. Otherwise your structure will become unstable or it becomes a free body and it cannot be solved by SAP2000. Applying joint restraints requires the following steps:

a.

- a. Click the Pointer Tool button (i.e.) in the Floating Toolbar.
- b. Click the joints which have the same restraints.
- c. From the **Assign** menu, choose the **Joint→ Restraints...** from the submenu. This will display the joint restraint dialog box.
- d. In this dialog box, choose appropriate restraint parameter. Then click **OK** to accept this assignment.
- e. Repeat steps a, b, c and d until you finish the restraint assignment.

PART II. Solving In this part SAP2000 will assemble and solve the global matrix. The following steps are needed:

1. From the **Analysis** menu, select **Set Option...** This will display the Analysis Option dialog box.
2. In this dialog box, check the available DOF. If you are analyzing a plane truss, check UX and UY, leave the UZ, RX, RY and RZ blank.
3. Click **OK** to accept what you choose.
4. From the **analysis** menu, select **Run**. This will display the **Save Model File As** dialog box.
5. In this dialog box, save the model under a filename. No extension is necessary.
6. Click the **OK** button, the analysis will begin. A top window is opened in which the various phases of analysis process are progressively reported. When the analysis is complete, the screen will display the message "ANALYSIS COMPLETE".
7. Click **OK** button in the top window to close it.

PART III. Postprocessing.

The main options in postprocessing are:

1.
 1. Displaying the deformed shape.
 2. Displaying the member forces.
 3. Printing the results.
 4. Designing the structural members and checking the safety of a design.
 5. Modifying the structure.

For simplicity, we just discuss the three fundamental options: displaying the deformed shape, displaying the member forces and printing results here.

1. Displaying the deformed shape.

After the analysis is complete, SAP2000 automatically displays the deformed shape of the model for the default load case, LOAD1, in the active display window. We can now display the deformed shape for another load case in one of the two view windows.

- a. Activate one of the two view windows by clicking anywhere inside that window.
- b. Click the display deformed shape button on the floating toolbar. This will display the deformed shape dialog box.
- c. In the drop down list in the load area of this dialog box, select the load case to be displayed, then click **OK** button. The deformed shape will show.

1. Displaying the member forces.

- a. From the **Display** menu, click the **Show element forces/stresses → frames**, this will display the member force diagram dialog box.
- b. In this dialog box, select the component which need to display (for truss, choose Axial force) in the **Component** area, and click **OK** button. The axial force diagram for the entire truss is displayed. By moving cursor to a specific location, we can read the values of the force at that point.

1. Printing the results.

- a. From **File** menu, select **Print Output Table...** In the display dialog box, click **OK** to accept the default setting. The detailed output results will be printed.
- b. From **File** menu, select **Print Input Table...** In the display dialog box, click **OK** to accept the default setting. The detailed input information will be printed.

You can also get the detailed results in another way. When we analyze a structure, by default, SAP2000 will create three output files: filename.out, filename.log and filename.EKO. The output file filename.out stores the output of your analysis. The output file filename.EKO stores the input information for this structure. The output file filename.log take all of the running information. These files are text files. You

can print these files using computer operating system. For example, we can print these files from Notepad. The steps are:

a.

- a. Open Notepad by double click the Notepad icon on the main window.
- b. From **File** menu, choose **Open**. This will display a standard Microsoft file selection dialog box.
- c. In this dialog box, choose the drive and subdirectory where your file is located.
- d. Click on the file name you want to display and print. (i. e. any one of filename.out, filename.EKO, or filename.log.)
- e. Click **OK** to terminate this dialog box. Your file will display by Notepad.
- f. Review the file to make sure your results are correct.
- g. From **File** menu, choose **print...** This will display the print dialog box.
- h. Click **OK** to accept the default print setting. Your file will print on background.
- i. Repeat steps b, c, d, e, f, g and h to print another file.
- j. Close Notepad by choosing **Exit** from the **File** menu.

[Subscribe \(Full Service\)](#) [Register \(Limited Service, Free\)](#) [Login](#)Search: [The ACM Digital Library](#) [The Guide](#)[Feedback](#) [Report a problem](#) [Satisfaction survey](#)

Real-time simulated earthquake motion of high rise structures

Full text [Pdf \(721 KB\)](#)**Source** [Annual ACM IEEE Design Automation Conference archive](#)
[Proceedings of the 7th workshop on Design automation table of contents](#)
San Francisco, California, United States
Pages: 35 - 46
Year of Publication: 1970**Author** [James](#) Principal Systems Engineer, Albert C. Martin and Associates, Planning/Architecture/Engineering, Los Angeles, California**Sponsors** IEEE : Institute of Electrical and Electronics Engineers
[ACM](#): Association for Computing Machinery
SHARE : SHARE**Publisher** ACM Press New York, NY, USA**Additional Information:** [abstract](#) [references](#) [index terms](#) [peer to peer](#)**Tools and Actions:** [Discussions](#) [Find similar Articles](#) [Review this Article](#)
[Save this Article to a Binder](#) [Display Formats: BibTex](#) [EndNote](#) [ACM Ref](#)

↑ ABSTRACT

The real-time simulation of a high rise structure during an earthquake represents a marked step forward in the structural design field. It affords a graphic display of the dynamic behaviour and characteristics of such a structure. The simulation can be presented on a CRT or by means of a 16 or 35 mm. movie and tailored for client or technical viewing. This paper discusses the application of the computer techniques involved in dynamically analyzing and simulating the seismic response of an actual 52 story steel-framed tower located in the Los Angeles area.

↑ REFERENCES

Note: OCR errors may be found in this Reference List extracted from the full text article. ACM has opted to expose the complete List rather than only correct and linked references.

1 Merchant, H.C., and Hudson, D.E., "Mode Superposition in Multi-degree-of Freedom Systems Using Earthquake Response Spectrum Data". *Bulletin of the Seismological Society of America*, Vol. 52, No. 2, April 1962, pp. 405-416.

2 Shepard, R., "Some Limitations of Modal Analysis in Seismic Design". *Bulletin of the New Zealand Society for Earthquake Engineering*, Vol. 2, No. 3, Sept. 1969, pp. 284-288.

3 Jennings, P.C., "Spectrum Techniques for Tall Buildings". *Proceedings of the Fourth World Conference on Earthquake Engineering*, Santiago, Chile. Jan. 1969

4 Martin, Albert C. and Associates, "Computer Program - ACMDYN". ACMA Program Library, Los Angeles, California. Co-operation of E.L. Wilson of the University of California, Berkley, in the program development, is acknowledged.

5 Martin, Albert C. and Associates, "Computer Program - EQGDMO". ACMA Program Library, Los Angeles, California.

6 Jennings, P.C., "Simulated Earthquake Motions". Proceedings of the Fourth World Conference on Earthquake Engineering, Santiago, Chile. Vol. 1, Jan. 1969, A-1, pp. 145.

↑ INDEX TERMS

Primary Classification:

J. Computer Applications

↪ **J.7 COMPUTERS IN OTHER SYSTEMS**

↪ **Subjects:** Real time

Additional Classification:

J. Computer Applications

↪ **J.2 PHYSICAL SCIENCES AND ENGINEERING**

↪ **Subjects:** Earth and atmospheric sciences

General Terms:

Theory

↑ Peer to Peer - Readers of this Article have also read:

- Data structures for quadtree approximation and compression

Communications of the ACM 28, 9
Hanan Samet

- A hierarchical single-key-lock access control using the Chinese remainder theorem

Proceedings of the 1992 ACM/SIGAPP Symposium on Applied computing
Kim S. Lee , Huizhu Lu , D. D. Fisher

- The GemStone object database management system

Communications of the ACM 34, 10
Paul Butterworth , Allen Otis , Jacob Stein

- Putting innovation to work: adoption strategies for multimedia communication systems

Communications of the ACM 34, 12
Ellen Francik , Susan Ehrlich Rudman , Donna Cooper , Stephen Levine

- An intelligent component database for behavioral synthesis

Proceedings of the 27th ACM/IEEE conference on Design automation
Gwo-Dong Chen , Daniel D. Gajski

[Terms of Usage](#) [Privacy Policy](#) [Code of Ethics](#) [Contact Us](#)

Useful downloads:  [Adobe Acrobat](#)  [QuickTime](#)  [Windows Media Player](#)  [Real Player](#)

REAL-TIME SIMULATED EARTHQUAKE MOTION
OF HIGH RISE STRUCTURES

James Lord
Principal Systems Engineer
Albert C. Martin and Associates
Planning/Architecture/Engineering
Los Angeles, California

SYNOPSIS

The real-time simulation of a high rise structure during an earthquake represents a marked step forward in the structural design field. It affords a graphic display of the dynamic behaviour and characteristics of such a structure. The simulation can be presented on a CRT or by means of a 16 or 35 mm. movie and tailored for client or technical viewing.

INTRODUCTION

This paper discusses the application of the computer techniques involved in dynamically analyzing and simulating the seismic response of an actual 52 story steel-framed tower located in the Los Angeles area. The tower was modelled mathematically and excited by several earthquakes. A dynamic analysis obtained the time history of the responses and ground motion. The combined response for the 1940 El Centro (N-S component) earthquake was then simulated in real-time on a CRT and the motion of the tower recorded on a 16 mm. movie. This seven minute movie portrays several sequences of the combined seismic response at scale factors noted in parenthesis, including;

1. Ground motion (x 120) plus actual building response for the first eight modes of vibration.
2. Combined ground and building motion of the first eight modes (x 120).
3. Coincidental modal responses of the first three modes of vibration (x 67), including ground motion. These individual modal responses were isolated and displayed as three separate images on each frame of the sequence.
4. An interrupt sequence of 3. Several stop frames have been introduced to illustrate certain dynamic characteristics.
5. Repeat of sequence 2.

The paper comprises:

A general discussion of the state of the art of dynamic analysis as related to buildings and building codes. Direct integration, normal mode, and modal

superposition - response spectra methods are reviewed;

. A description of the analysis phase, including a detailed look at the computer programs used;

. An outline of the post-analysis procedures, discussing the techniques involved in simulating the building motion first on the CRT and then on film. The outline includes a brief description of the software package and the hardware configuration utilized.

. An evaluation of the analysis and simulation is made. The paper contains several before-after portrayals of the maximum responses of the structure, illustrating how certain undesirable dynamic characteristics were in fact controlled.

DYNAMIC ANALYSIS - GENERAL REVIEW

Basically there are three categories or types of dynamic analysis.

1. Direct integration of the general equations of motion.
2. Normal mode analysis.
3. Response spectra techniques.

1. The direct integration method is a perfectly general technique. It is applicable to coupled, uncoupled, elastic or inelastic systems vibrating under any loading configuration. For a multi-degree of freedom system, subject to support motion, we have in matrix notation;

$$[M] \{ \ddot{y} \} + [C] \{ \dot{u} \} + [K] \{ u \} = 0$$

Where

$[M]$ = diagonal matrix of successive story masses for a lumped mass analysis.

$[C]$ = damping coefficient matrix

$[K]$ = lateral stiffness matrix

$\{ \ddot{y} \}$ = absolute acceleration vector

$\{\ddot{u}\}, \{\dot{u}\}, \{u\}$ = relative to base lateral acceleration, velocity and displacement vectors respectively

Since $\{\ddot{y}\} = \{\ddot{u}\} + \ddot{y}_s$ by definition

Back substituting in the general equation of motion

We have $[M] \{\ddot{u}\} + [C] \{\dot{u}\} + [K] \{u\} = -[M] \ddot{y}_s$

Where

$\{M\}$ = vector of successive story masses

$\ddot{y}_s(t)$ = time varying ground acceleration

Direct integration of this latter equation of motion yields the lateral accelerations and displacements story by story from which all joint deformations and rotations, story shears and overturning moments and member stress levels can be obtained.

2. The normal mode method is a limited technique. It is applicable only to linear elastic systems vibrating under the action of loads having a common time function. These limitations allow any close or far coupled system to be uncoupled and treated as several independent single degree of freedom systems. This is to say that all the modes of vibration are independent of each other. For the n^{th} mode, the modal equation of motion is;

$$\ddot{A}_n + 2W_n C_n \dot{A}_n + W_n^2 A_n = -\ddot{y}_s(t) \sum_{r=1}^j \theta_{rn} M_r$$

Where

W_n = natural circular frequency of vibration for the n^{th} mode

C_n = fraction of critical damping in the n^{th} mode

$\ddot{A}_n, \dot{A}_n, A_n$ = modal amplitudes of acceleration, velocity and displacement for the n^{th} mode, relative to base.

$\ddot{y}_s(t)$ = time varying ground acceleration
 M_r = r^{th} mass

θ_{rn} = normalized characteristic displacement at the r^{th} mass for the n^{th} mode.

j = total number of masses (stories) in structure.

By integrating the above modal equation

for each of the modes to be considered and summing each response parameter A_n, \dot{A}_n, A_n in normal coordinates, the response parameters \ddot{y} , \dot{y} & y can be obtained by the modal relationship $\ddot{y} = \Phi \ddot{A}_n$ etc.

Where

Φ = square matrix containing all normalized eigenvectors such that the n^{th} column corresponds to the n^{th} mode.

Hence the remaining essential response parameters can be computed directly.

3. A response spectrum is a plot of the maximum value of any response parameter against the period of vibration for a linear elastic single degree of freedom system. Figure 1 shows a typical response spectrum for the pseudo-velocity @ 5% damping for the A1, A2 and the 1940 El Centro (NS component) earthquakes. Relative displacements and hence stress levels can be expressed in terms of the velocity spectrum through simple relationships. For simple structures such as elevated water tanks, response spectra can be used directly to obtain the structural response. For complicated structures, in which the contribution of higher modes of vibration are significant, the accurate prediction of maximum values of any response parameter is extremely difficult. This is because the time at which the maximum response occurs is not preserved in the spectral curves and since the maximum values of the individual modes do not necessarily occur simultaneously, the phase relationship between the maxima are lost, and hence the exact way in which the various modes combine cannot be determined. Several approximate methods of mode combination have been suggested, ranging from simply taking the sum of the absolute values (Biot 1943) to taking the weighted average of the square root of the sum of the squares of the maximum individual modal responses. (Hudson & Merchant 1962)¹. The former method obviously would give the worst case and merely sets an upper bound. Even the refinements of the latter method have some limitations as pointed out by Shepard². It should be noted that the use of any response spectra requires an accurate knowledge of the natural periods and mode shapes.

The state of the art of dynamic analysis as related to buildings in particular, is one of fast development. Wind dynamics and earthquake engineering have become major considerations to the structural engineer. He is aware of the inadequacies of the code static analysis

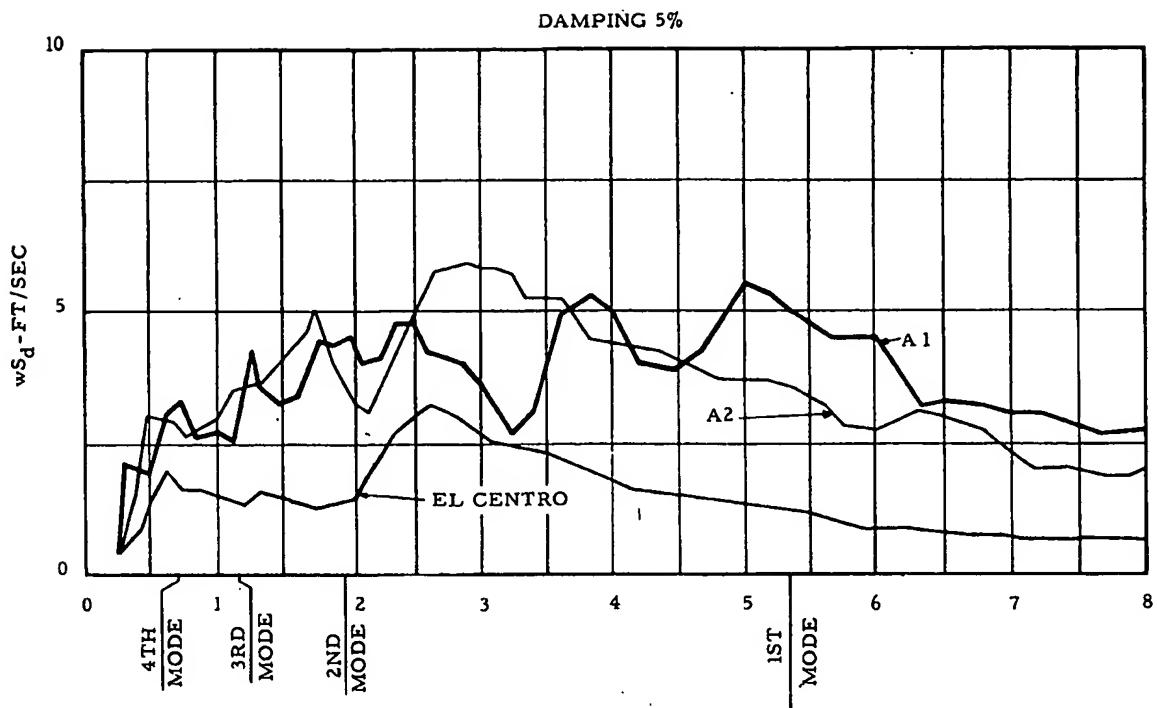


FIGURE 1
VELOCITY SPECTRUM

approaches. The large digital computer is now his tool in solving the dynamics problem. He is able to utilize all the three types of dynamic analysis outlined earlier. Computer technology has stimulated the use of the direct integration and normal mode analysis methods. In general, response spectra techniques are not used to predict responses, but serve to correlate computed elastic response calculations. They are used, for instance, to determine the desired length of earthquake record to be used in a normal mode or step-by-step integration solution. Spectral techniques also have been very effective in establishing trends in linear responses of tall buildings to earthquake motion³.

ANALYSIS PHASE

General

The generation of the time history of the building response including individual modal participation and the ground motion history comprised the analysis phase. Reference to Figure 2a will indicate that two computer programs "ACMDYN"⁴ and "EQGRMO"⁵ and a 131K Univac 1108 hardware configuration were used to implement this phase.

Description of Structure

The 52 story, steel-framed tower was modelled mathematically and excited by several earthquakes. The simulation was carried out in the longitudinal direction of the building for the 1940 El Centro (N-S component) earthquake. The natural periods of vibration for the first eight modes in this direction were: 5.38, 1.98, 1.19, 0.85, 0.66, 0.54, 0.46, 0.40 seconds respectively. The structure remained entirely elastic throughout the thirty second duration of the earthquake. This was not surprising as the tower was sized using the 1940 El Centro (N-S component) as the design earthquake at yield level. A typical floor plan of the tower is included as Figure 4. Figure 3 represents a transverse section through the tower. Plan dimensions of the tower are 220 ft x 120 ft. The height above the surrounding plaza is 650 ft. In the longitudinal direction only the exterior frames participate in resisting lateral forces on the structure. Three dimensional or box action was considered in which the end wall frames participate in resisting the overturning moments induced by lateral loads in the side wall frames. A suitable linking technique was devised at the corner column to permit the transfer

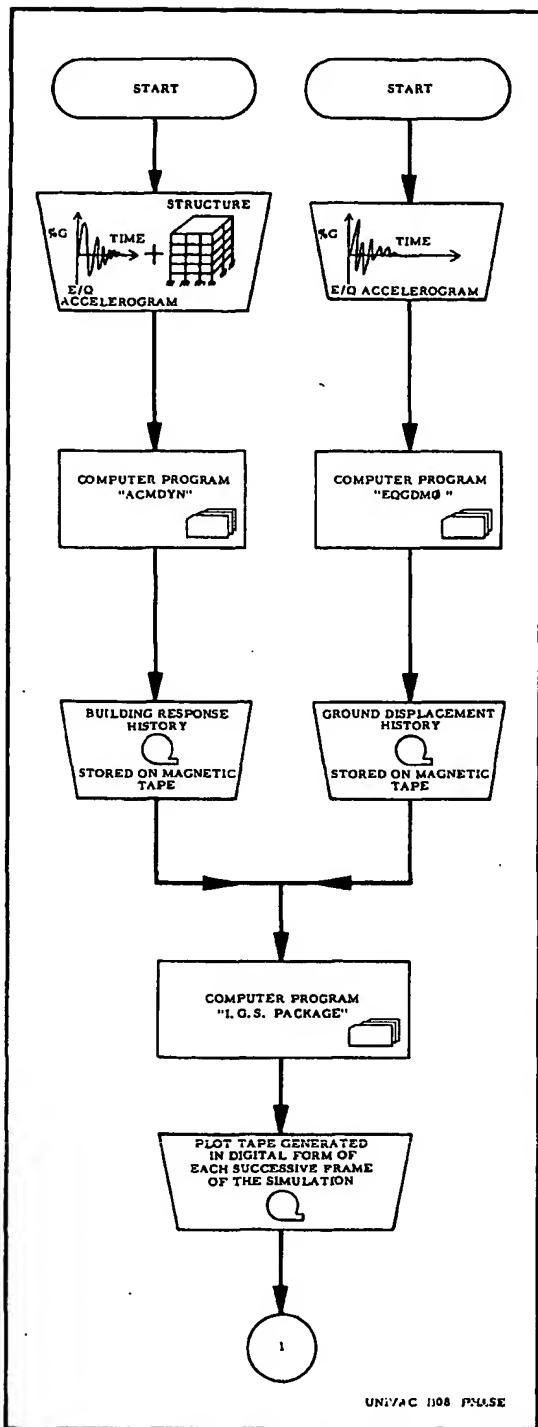


FIGURE 2a - ANALYSIS PHASE

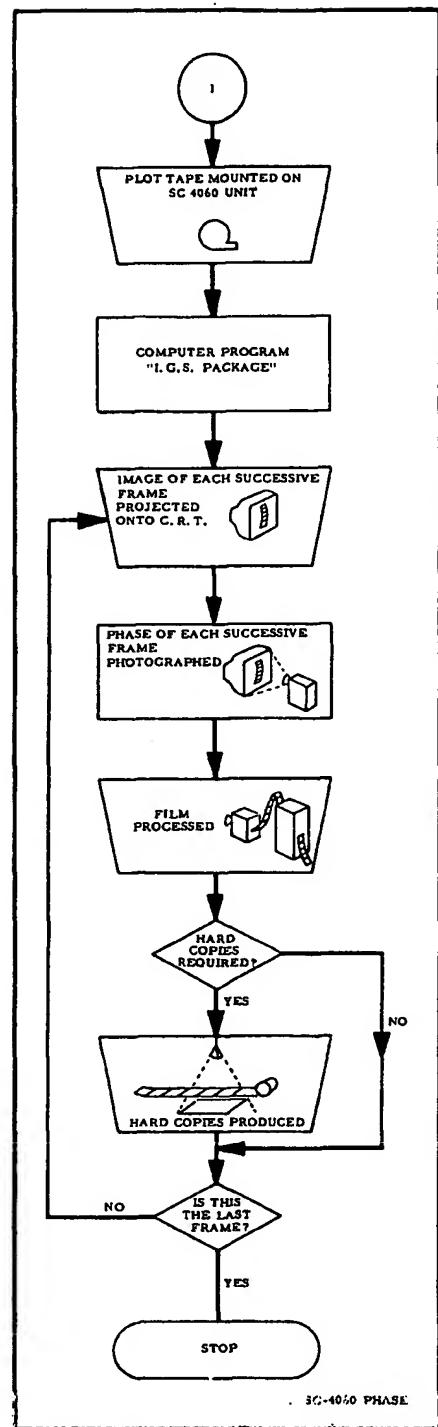


FIGURE 2b - SIMULATION PHASE

FIGURE 2 PHASE FLOW CHART OF THE ANALYSIS AND SIMULATION TECHNIQUES

of vertical load into the transverse end wall frames without exerting any fictitious rotational restraints or transmitting any lateral shears around the corner. Owing to symmetry, only one-quarter of the structure was analyzed. The transverse direction was modelled identically except for the inclusion of six-two bay interior moment resisting steel frames below the 28th floor. The stiffness parameters of the structure were evaluated on a center line to center line basis.

Excitation

The seismic excitation was applied along each of the major axes of the building in turn. A total of nine earthquake records were used to induce varying degrees of vibration in the structure. All of the simulated earthquake motions (Jennings, Housner and Tsai 1968)⁶, modelling ground shaking of the order expected from earthquakes of magnitude 4.5 through 8.5 on the Richter Scale were utilized, plus the 1940 El Centro record (7.2 magnitude).

For the simulation run, the time interval of integration of the modal equations of motion was set at 1/40 second, thus generating histories of the

relative lateral displacements of the 1st, 2nd and 3rd modes individually and for all first 8 modes combined at each floor level at the same time interval. These four time histories were then sequentially stored on magnetic tape.

Program Description

An abstract of the computer program "ACMDYN"⁴ follows. It is included to describe the extensive capabilities of the program.

TITLE: ACMDYN
TYPE: Analytical
COMPUTER: Univac 1108 (131^K core),
 CDC 6600 (131^K core)
LANGUAGE: Fortran V
CAPACITY: 70 stories, 30 bays, unlimited
 number of different frames

PURPOSE AND DESCRIPTION

"Elastic, two-dimensional dynamic lumped mass analysis of steel or concrete, braced or unbraced orthogonal frames, with or without shear walls, using the normal mode method. (2n+1) degrees of freedom are allowed at each story, where n = no. of joints in frame @ any story. Three dimensional or box action may be considered. Beams and/or columns may be either prismatic or non-prismatic, pinned

52 STORY TOWER

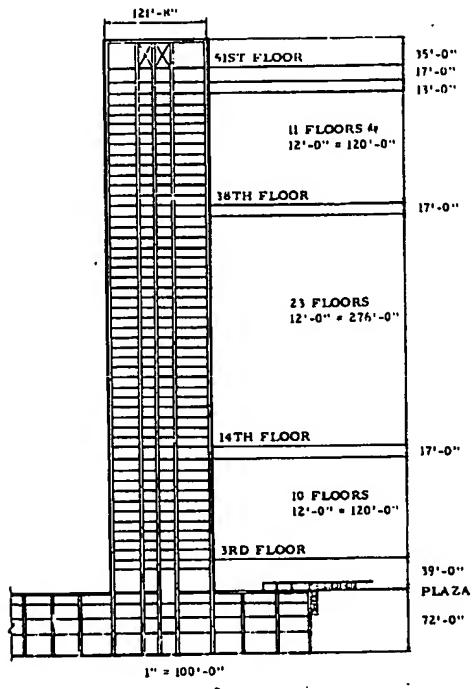


FIGURE 3
 TRANSVERSE SECTION

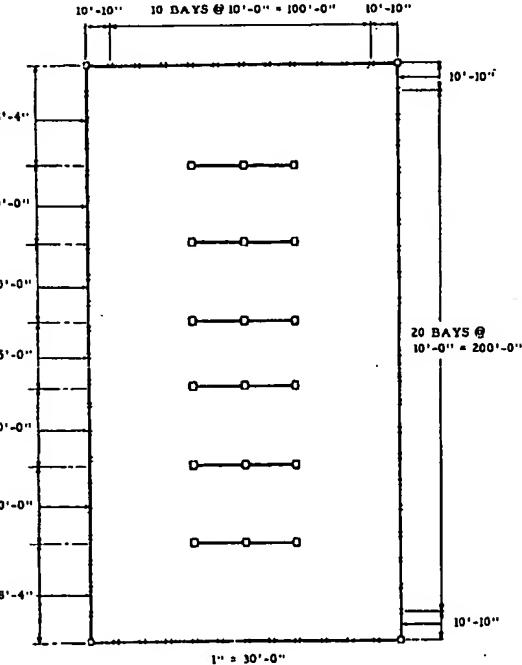


FIGURE 4
 TYPICAL FLOOR FRAMING

or restrained at either or both ends. Diagonal bracing is limited to pin-ended members. Axial column shortening, shear deformations and finite joint size effects are considered. Allows for lateral exterior restraints to be applied at any story. Obtains time history responses for absolute floor accelerations, drift, inter-story drift, story shears and story overturning moments and maximizes on each response. Constant damping in each mode. Determines periods and mode shapes of structure. Establishes individual percentage modal contributions of each of the maximum responses. Determines maximum member loads and stress levels. Establishes maximum vertical displacements and rotations at each joint and determines the maximum slope coefficient for each girder. Handles multiple earthquake per run. Any number of problems may be solved in each run. Plot options for plotting maximum responses are incorporated. Evaluates steel tonnage when applicable."

The lateral displacements computed by "ACMDYN" are those that are relative to base. To achieve a meaningful simulation, it was necessary to obtain and portray the absolute lateral displacement histories for each story. This was accomplished by generating the time history of the ground displacement at the 1/40 second interval and algebraically summing it to the corresponding time history for the relative displacements. As Figure 2 indicates, the computer program "EQGRMO"⁵ performed the necessary integration procedures. By taking the variable interval earthquake accelerogram of the 1940 El Centro (N-S component) earthquake and executing successive integrations to yield first the ground velocity and then the ground displacement history, a magnetic tape was generated, storing the base motion record at a 1/40 second time interval. The program incorporates an integration routine based on a combination of Simpson's and Newton's 3/8 rules. A plot was obtained on a constant line ink plotter to verify the accuracy of the base motion.

SIMULATION PHASE

General

The simulation was effected by utilizing the Integrated Graphics Software (I.G.S.) System in conjunction with a Stromberg-Carlson 4060 Computer recorder. The I.G.S. package is a comprehensive system comprising a library of subroutines to provide various display functions. It is Fortran-oriented, although it may be called from assembly

language, Cobol, PL/1 or any other symbolic language compatible with the subroutine linkage conventions. It was developed as a co-operative effort between Stromberg Data Graphics Incorporated and the Rand Corporation. In 1968, the Naval Weapons Center at China Lake, California, released a Univac 1108 version that included additional polar routines. This version allows the 1108 to accept data and graphic formatting instructions, to perform necessary scaling, conversions and translations, to create meta-language output and to write it onto magnetic tape or other output device. The meta-language information is then read by the product control unit (a DDP ST6 computer) in the S-C 4060 for conversion to the sequence of hardware commands required to produce the CRT graphic display, either a 16 mm. or 35 mm. film record and a hardcopy print of any or all frames as desired. Figure 2b represents a diagrammatic flowchart of this simulation phase.

Animation Technique

The basic technique employed was one of animation. A plot of the tower configuration was generated and stored in meta-language on a magnetic tape, at a given point in time. The plot resulted from the superposition of the coincident ground displacement and the building response for each story. Each plot then constituted a frame in the final movie. The tower was represented as a frame and not simply as a single line. Floor levels were described and plotted. By programming the advance of the film, successive frames were generated for each point in time. The time interval between frames was selected as 1/40th of a second. This was done to correlate the time interval between frames and the speed of a movie projector. At normal speeds, 24 frames/second are projected on a screen. By achieving this correlation, the frequency of the tower vibration was preserved in the screen image of the tower. This means that the motion of the image is in real-time.

Purpose and Scale Factors

Prime consideration during development of the simulation was devoted to the purpose for which the simulation was intended and also to the scale factor involved. In this instance, it was desirable to tailor the simulation for client, technical and general viewing. As such, several sequences were compiled using various scale factors. For technical presentations it was deemed important to have the motion of the screen

image reflect the actual motion of the 650' high building.

To achieve this, a magnification factor of 120 was required to be applied to the horizontal scale. Expressed in different terms a reduction scale of 120 was applied to the vertical scale. This was based on a 5' high image. When projected to this image height, the lateral deflections, velocities, and accelerations are exactly those of the actual tower structure. This situation is represented by sequences 2 and 5. For client viewing, sequence 1 shows the base motion scaled on the order of the actual ground motion for a projected image five feet in height. The building distortion is scaled to the building size which on this size image is of the order of 1/120 of the actual building dimensions. This is intended to give the viewer the feel of the actual ground motion and yet show that in comparison with the building size, the lateral building distortions are barely perceptible.

Another sequence (sequence 3) was devised to illustrate the concept of modal participation. The complex distortion of a building when elastically vibrating under any set of dynamic forces represents the summation of several simple curves or mode shapes. Associated with each mode shape is a natural period of vibration. The degree to which these independent modes are excited depends on the particular forcing function. At any given point in time at a given story, the lateral displacement of the tower can be identified as having contributions from all of the participating modes. This was illustrated by displaying the first three modes of vibration on a single frame of the movie. The magnification factor was reduced to 67 for this sequence, simply to accommodate the three separate images. Each image depicts a separate coincidental mode of vibration. This entire sequence was repeated as sequence 4, to include several interrupt features. By generating a number of identical frames on the plot tape, a stop-action feature was introduced. Four such features were incorporated as shown in Figure 5.

Figure 5a indicates the largest relative to base 3rd mode contribution. This occurs on the 13th floor at 3.5 seconds.

1st mode contribution	+2.8"	(+350%)
2nd mode contribution	-1.1"	(-138%)
3rd mode contribution	<u>-0.9"</u>	(-112%)
$\sum (1+2+3)$	+0.8"	

The actual displacement relative to base, at this time, of all 1st eight modes = +0.8". This does not indicate that higher modes are negligible in themselves, rather that their summation is. Base displacement at 3.5 seconds = -6.0"

Figure 5b shows the modal participation at 4.0 seconds. The largest 1st mode displacement relative to base occurs at this time and, of course, occurs at the roof. The maximum roof deflection and the maximum base overturning moments also occur at this time. This illustrates that both of these parameters are essentially governed by the 1st mode response.

Breakdown:

1st mode contribution	+15.8"	(+82%)
2nd mode contribution	+ 2.7"	(+14%)
3rd mode contribution	<u>+ 0.7"</u>	(+ 4%)
$\sum (1+2+3)$	+19.2"	

Base displacement at 4.0 seconds = -3.2"

Figure 5c illustrates the largest 2nd mode participation. Occuring at 6.4 seconds at the 21st floor, the combined displacement of the 1st, 2nd and 3rd modes (rel. to base) is -5.4" which again approximates the displacement considering the first 8 modes of vibration.

Breakdown:

1st mode contribution	-2.9"	(+54%)
2nd mode contribution	-2.7"	(+50%)
3rd mode contribution	<u>+0.2"</u>	(- 4%)
$\sum (1+2+3)$	-5.4"	

Base displacement at 6.4 seconds = -5.4"

Figure 5d depicts the opposing nature of the 1st and 2nd modes at the roof level. Occuring at 9.6 seconds the breakdown is:

1st mode contribution	+8.2"	(+152%)
2nd mode contribution	-2.0"	(- 37%)
3rd mode contribution	<u>-0.3"</u>	(- 5%)
$\sum (1+2+3)$	+5.9"	

Base displacement at 9.6 seconds = -8.8"

Other maxima of the response are;

1. Maximum absolute displacement of roof occurs at 2.4 seconds = 23.4"
2. Maximum ground displacement occurs at 2.5 seconds = -11.4"
3. Final ground displacement = -7.5"

The ground displacement history depends on the particular version of the El Centro earthquake record used and on the type of

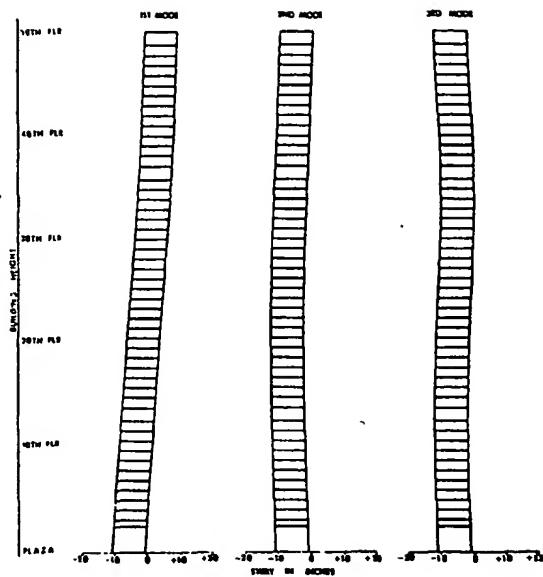


FIGURE 5A
MODAL PARTICIPATION AT 3.6 SECS.

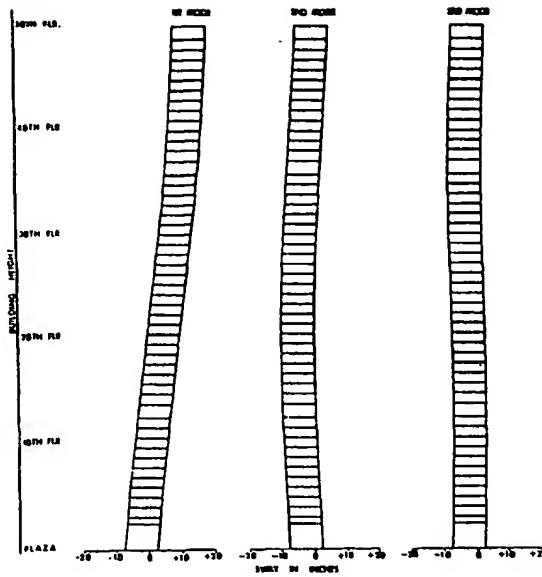


FIGURE 5B
MODAL PARTICIPATION AT 4.0 SECS.

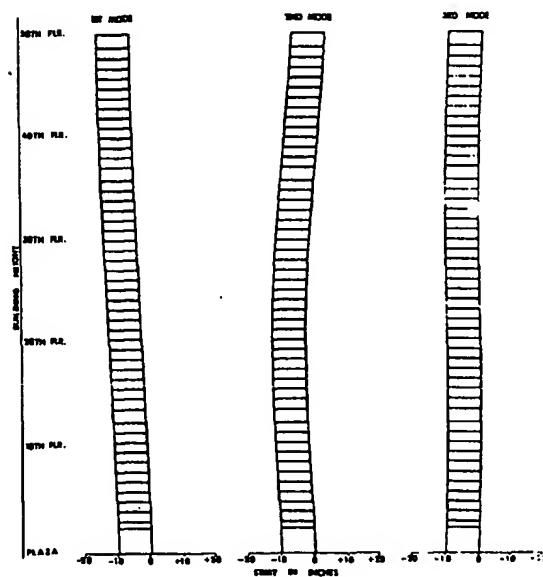


FIGURE 5C
MODAL PARTICIPATION AT 6.4 SECS.

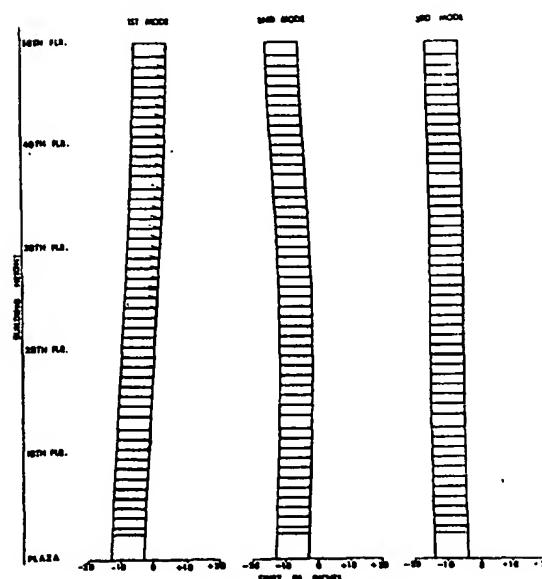


FIGURE 5D
MODAL PARTICIPATION AT 9.6 SECS.

FIGURE 5
SIMULATED EARTHQUAKE RESPONSE OF A 52 STORY TOWER
FOR THE MAY 1940 EL CENTRO (NS COMPONENT) GROUND MOTION.

baseline correction applied. For the purposes of this simulation the accuracy of the digitization or the baseline correction was not considered significant.

To complete the movie, a repeat sequence (sequence 5) was added showing the combined motion of ground and tower at the same scale factor of 120. Each configuration in Figure 5 was drawn directly on a constant line ink plotter from stored data on the plot tape.

EVALUATION OF THE ANALYSIS AND SIMULATION

Benefits to the Client

The benefits of this type of analysis and simulation to a client are significant. Some of these are enumerated as follows:

- . Identifies the degree to which a structure will adequately resist earthquake shaking.
- . Determines essential information which can form the basis of an earthquake risk analysis. Such a risk analysis can result in possible savings in earthquake insurance premiums. A client can either negotiate lower insurance rates or alternatively limit his coverage to some value less than the value of his total investment. In general, an earthquake risk analysis will indicate the possibility, or otherwise, of either partial or total collapse, based upon the maximum possible earthquake ground shaking that authorities believe could occur at the building site. It will also identify damage cost estimates for various probabilities of ground motions associated with a given period of time. It may, if the situation permits, establish an upper monetary limit of the damage, usually excluding fire risk, that may be sustained by the building concerned.
- . Establishes potential problem areas within a structure, possibly affecting location of equipment. For example, computer installations, sensitive instrumentation, etc.
- . Allows a safer structure to be designed for little or no extra cost.
- . Produces a structure that has been designed to reduce the level of human discomfort during a severe earthquake.
- . Provides a visual graphic assurance of the stability of a client's structure during any earthquake motion.

Benefits to the Structural Engineer

The use of this approach results in many benefits to the structural engineer. These advantages are summarized below:

- . Affords a check on manual sizing calculations. Figure 6 has been included to illustrate this. The two curves represent a before-after portrayal of the variation of the maximum drift coefficient over the height of the tower for the 1940 El Centro earthquake. The large peaks in the dashed curve at the 14th and 38th floors indicate sizing errors in the vicinity of the mechanical equipment floors. These errors, previously undetected, were graphically displayed. The solid curve represents the final corrected plot of the drift coefficient variation.
- . Points out the essential dynamic characteristics of the structure, for example it detects the presence or otherwise of whiplash effects.
- . Allows the engineer to control undesirable responses. To illustrate this, Figure 7 represents a plot of the maximum lateral absolute accelerations, which can be considered as a measure of the human response, for each story in the structure, when vibrating under the influence of the 1940 El Centro earthquake. The excessive acceleration of the 21st floor shown in the dashed curve, is due to a large second mode contribution. Final tailoring of the structure resulted in the elimination of this peak value as indicated by the solid line. The prime reason it was deemed necessary to correct this situation was because of its consistent appearance in most of the acceleration responses for other earthquake excitations.
- . Enables the engineer to tailor his structure to a consistent uniform strength over the full height of the structure.
- . Identifies the degree of drift control achieved for different levels of earthquake intensity, thereby establishing clearance requirements for exterior solid panelling etc.
- . Permits the engineer to design, for little or no extra cost in materials, a structure to withstand, adequately, earthquake forces far in excess of code minimums.
- . Allows the engineer to grasp the concept and the degree of modal participation in an elastic dynamic response.

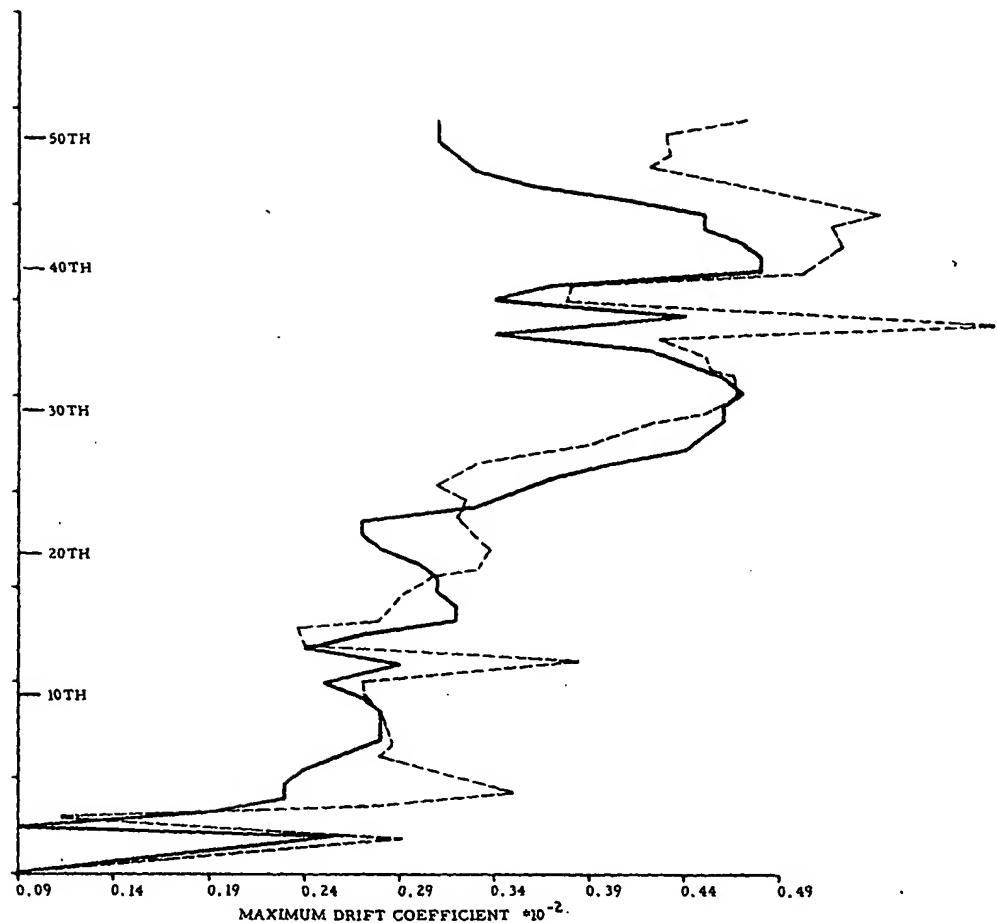


FIGURE 6
DRIFT COEFFICIENT

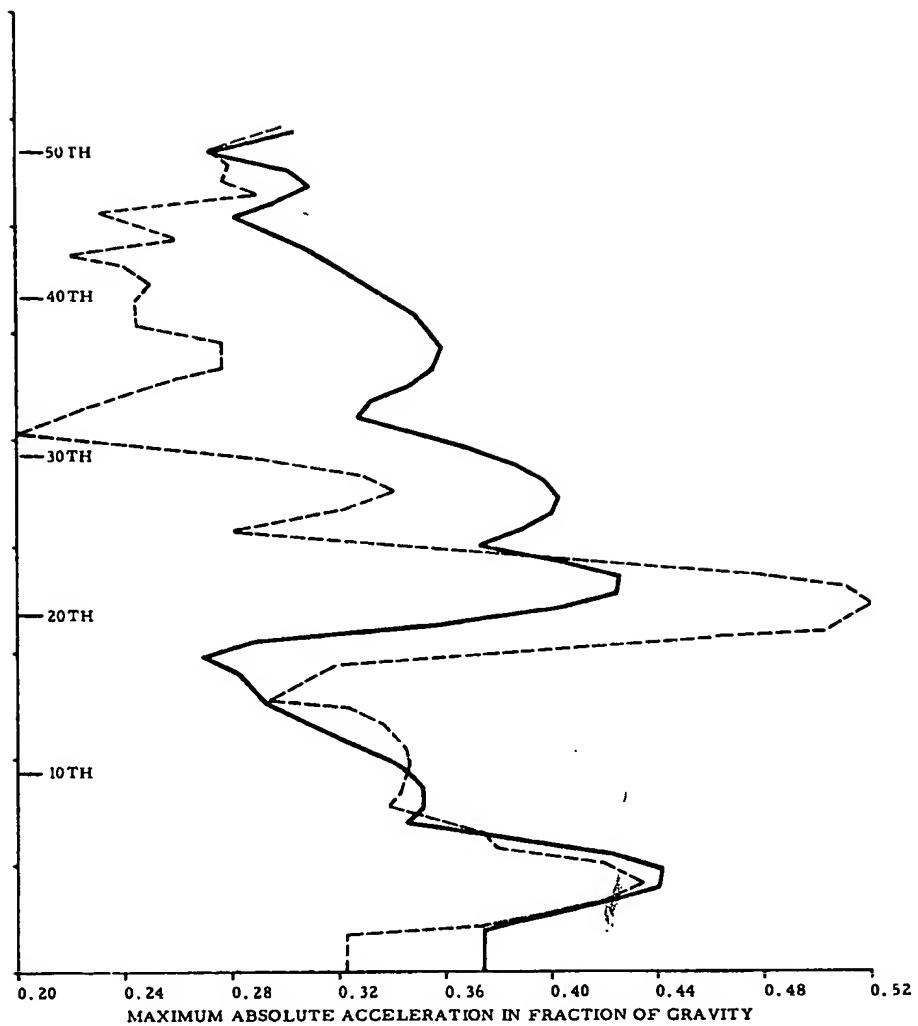


FIGURE 7
ACCELERATIONS

Enables the engineer to get a visual "feel" for his structure as well as a numeric one.

CONCLUSIONS

The techniques involved in both the analysis and simulation phases as described in this paper are well established and the necessary computer software packages to implement them, are generally available. The availability of, and easy access to the large digital computer has enhanced the practicability of these analysis and simulation techniques, the application of which could possibly be extended to the field of wind dynamics for high rise structures. The cost factors involved are not exorbitant and are not, in general, sufficient to prevent their frequent use. Thus, considering the advantages enumerated in the evaluation summary, the use of these techniques in the seismic design of all structures is to be recommended.

REFERENCES

- 1) Merchant, H.C., and Hudson, D.E., "Mode Superposition in Multi-degree of Freedom Systems Using Earthquake Response Spectrum Data". Bulletin of the Seismological Society of America, Vol. 52, No. 2, April 1962, pp. 405-416.
- 2) Shepard, R., "Some Limitations of Modal Analysis in Seismic Design". Bulletin of the New Zealand Society for Earthquake Engineering, Vol. 2, No. 3, Sept. 1969, pp. 284-288.
- 3) Jennings, P.C., "Spectrum Techniques for Tall Buildings". Proceedings of the Fourth World Conference on Earthquake Engineering, Santiago, Chile. Jan. 1969
- 4) Martin, Albert C. and Associates, "Computer Program - ACMDYN". ACMA Program Library, Los Angeles, California. Co-operation of E.L. Wilson of the University of California, Berkley, in the program development, is acknowledged.
- 5) Martin, Albert C. and Associates, "Computer Program - EQGDMO". ACMA Program Library, Los Angeles, California.
- 6) Jennings, P.C., "Simulated Earthquake Motions". Proceedings of the Fourth World Conference on Earthquake Engineering, Santiago, Chile. Vol. 1, Jan. 1969, A-1, pp. 145.

Organization _____ Biogran 101 DH
U. S. DEPARTMENT OF COMMERCE
COMMISSIONER FOR PATENTS

ALEXANDRIA, VA 22313-1450
IF UNDELIVERABLE RETURN IN TEN DAYS

OFFICIAL BUSINESS

AN EQUAL OPPORTUNITY EMPLOYER

<input checked="" type="checkbox"/>	REASON CHECKED
<input type="checkbox"/>	Owned, Len't Access
<input type="checkbox"/>	Unable to be read
<input type="checkbox"/>	Unknown
<input type="checkbox"/>	Uncertified
<input type="checkbox"/>	Used
<input type="checkbox"/>	No Such Street
<input type="checkbox"/>	Insufficient Address

RECEIVED
MAR 8 2006
MSPL/OMA/CL/ CENTER

**This Page is Inserted by IFW Indexing and Scanning
Operations and is not part of the Official Record**

BEST AVAILABLE IMAGES

Defective images within this document are accurate representations of the original documents submitted by the applicant.

Defects in the images include but are not limited to the items checked:



BLACK BORDERS



IMAGE CUT OFF AT TOP, BOTTOM OR SIDES



FADED TEXT OR DRAWING



BLURRED OR ILLEGIBLE TEXT OR DRAWING



SKEWED/SLANTED IMAGES



COLOR OR BLACK AND WHITE PHOTOGRAPHS



GRAY SCALE DOCUMENTS



LINES OR MARKS ON ORIGINAL DOCUMENT



REFERENCE(S) OR EXHIBIT(S) SUBMITTED ARE POOR QUALITY



OTHER: _____

IMAGES ARE BEST AVAILABLE COPY.

As rescanning these documents will not correct the image problems checked, please do not report these problems to the IFW Image Problem Mailbox.